Adjoint Shape Optimization for Aerospace Applications

Franklyn J. Kelecy
ANSYS, Inc.

Advanced Modeling & Simulation (AMS) Seminar Series
NASA Ames Research Center, April 8, 2021

Ansys

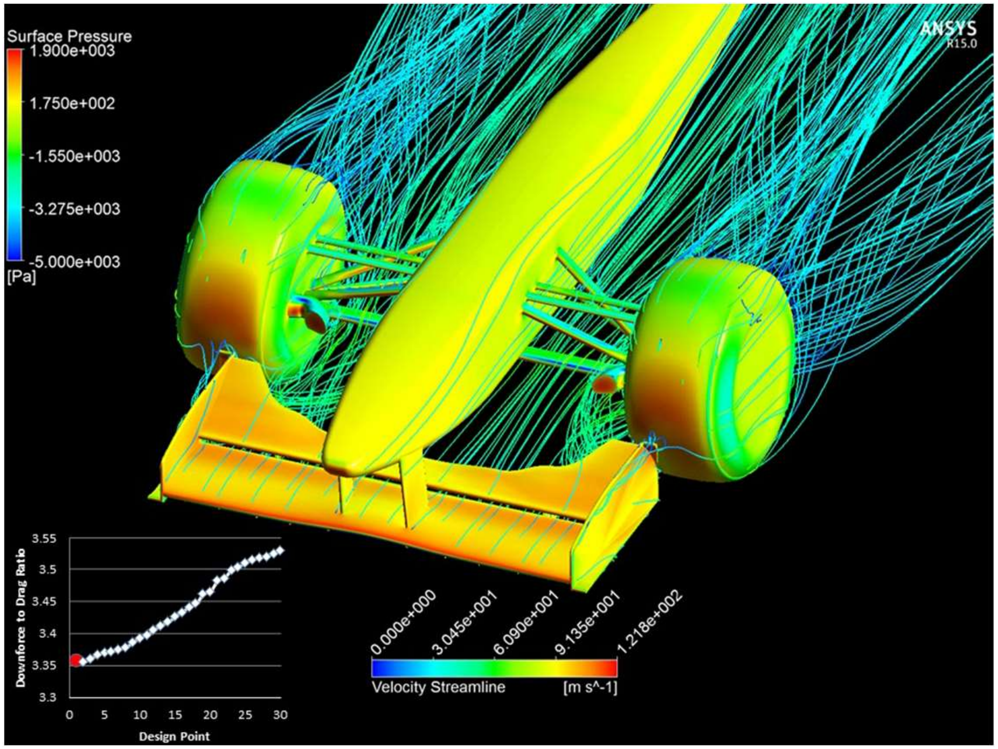
Agenda

- Introduction to Adjoint Methods for CFD
- Adjoint Theory Overview
- Adjoint Solver in Ansys Fluent
- Gradient-Based Optimizer
- Shape Optimization Examples
  - Simple cases (flow and heat transfer)
  - Supersonic wind tunnel
  - Hypersonic aerospike
- Summary + Questions

Introduction to Adjoint Methods for CFD

What is the Adjoint Method?

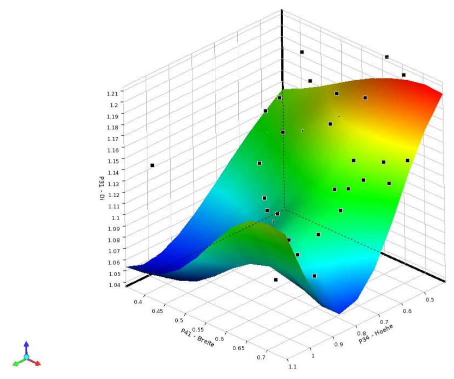
- The Adjoint Method is a specialized mathematical tool that extends the scope of a CFD solution by providing detailed sensitivity data for the performance of a fluid system subject to specific boundary conditions.
- The Adjoint Solver can be used to compute the derivative of an engineering quantities with respect to all inputs for the system. This includes the flow geometry!
- Consequently, it can be used to guide intelligent design modifications for shape optimization of any geometric feature in the computational domain.
- There are many more uses of Adjoint methods in CFD, but we will focus on shape optimization in this presentation.



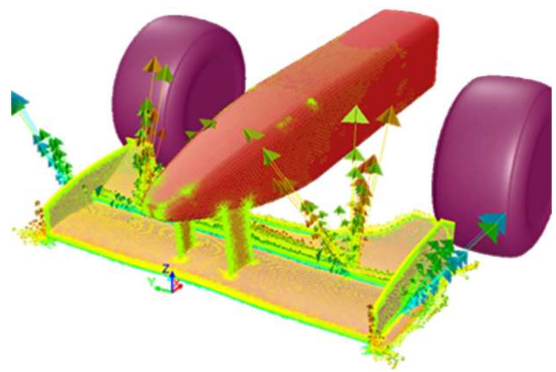
Adjoint-based shape modification of an automotive front spoiler

Parametric Design Optimization vs. Adjoint Optimization

- **Parametric Design Optimization**
  - Finds optimal operating conditions for given shape which can be described by a finite number of parameters.
  - Parametric behavior is determined by brute force (Design Of Experiments)
    - Computationally expensive as the number of parameters increases



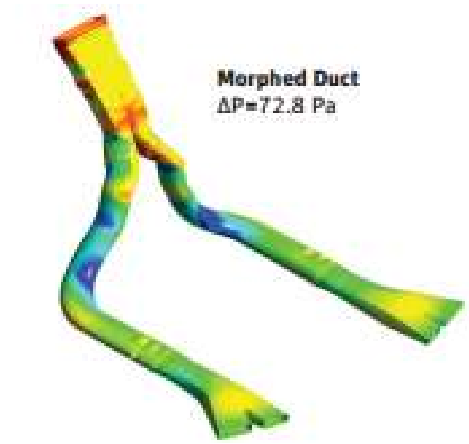
- **Adjoint Optimization**
  - Finds an optimal shape for a given operating condition
  - Derive the optimal shape from a baseline CFD flow calculation.
    - Smart design decisions possible with low investment of computation time
    - **No parameterization of the geometry is necessary!**



Example I

- **Rear Cabin Automobile HVAC Duct - Minimize Total Pressure Drop**

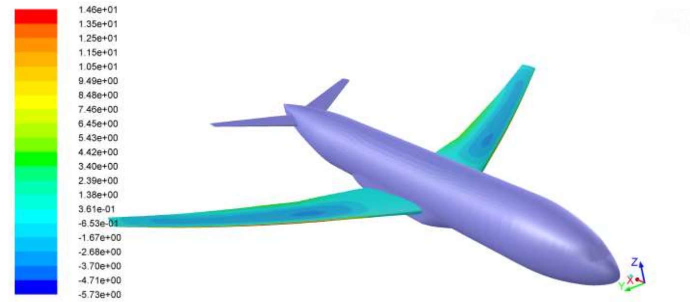
33% reduced pressure drop

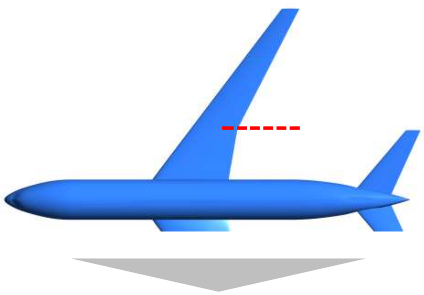


- Small geometry changes determined by Adjoint optimization / mesh morphing results in significant performance improvement!
  - This is a consequence of node displacements for specified portion of domain.

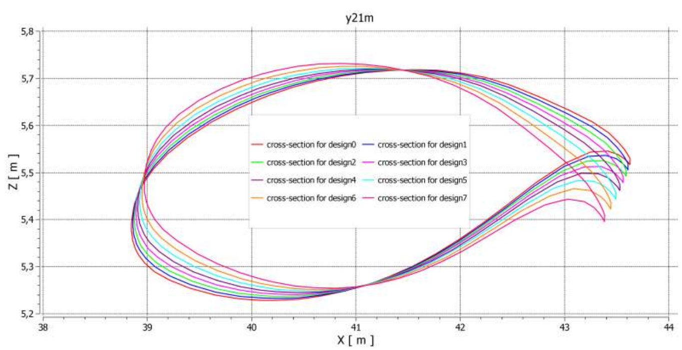
Example II

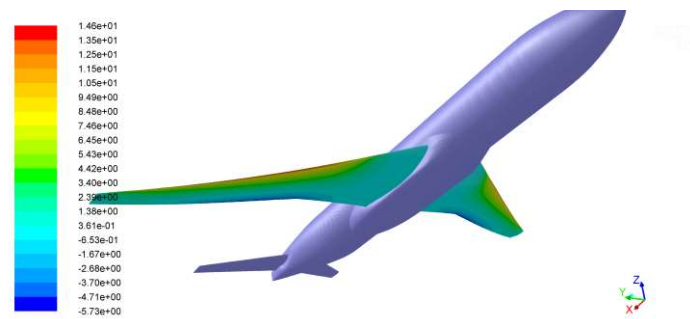
- **Airplane – full scale - Maximize Lift/Drag**
  - Optimize shape of the wing to increase lift to drag ratio
  - Again, just small changes result in a significant improvement…



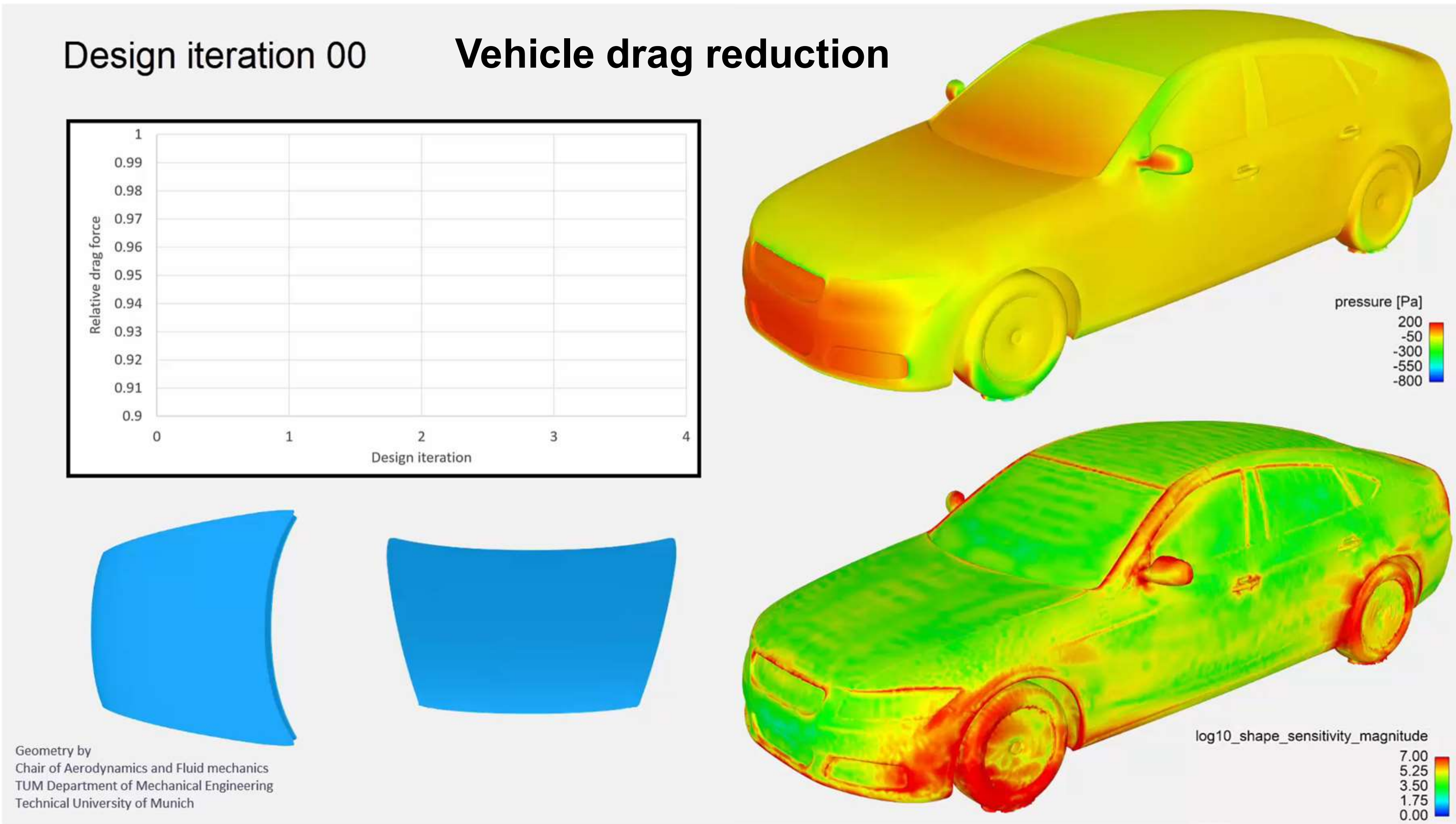


55% increased lift





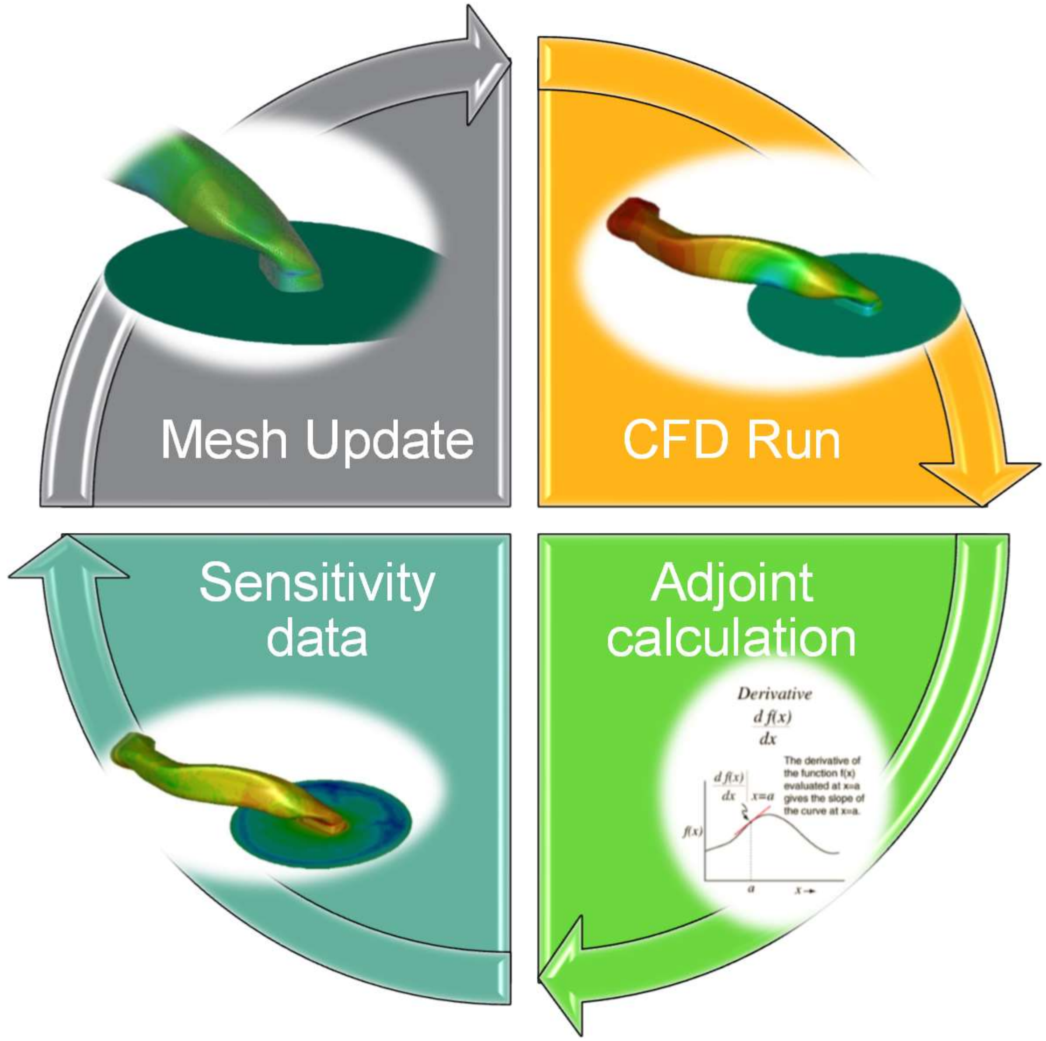
Example III


Design iteration 00
Vehicle drag reduction
Relative drag force
1
0.99
0.98
0.97
0.96
0.95
0.94
0.93
0.92
0.91
0.9
0
1
2
3
4
Design iteration
pressure [Pa]
200
-50
-300
-550
-800
log10_shape_sensitivity_magnitude
7.00
5.25
3.50
1.75
0.00
Geometry by
Chair of Aerodynamics and Fluid mechanics
TUM Department of Mechanical Engineering
Technical University of Munich

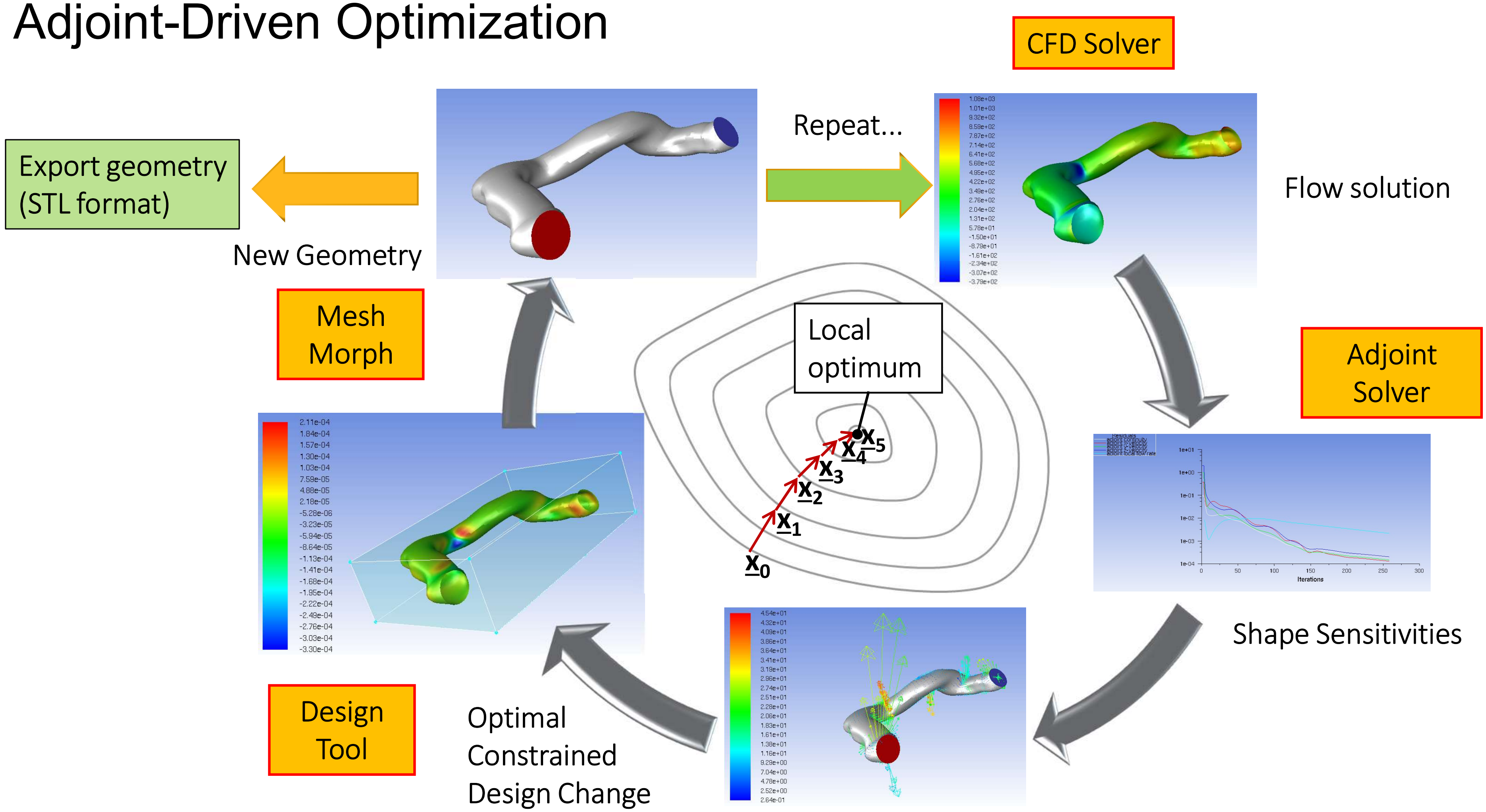
Overview of the Workflow

- The workflow can be viewed as a four-step process
  1. CFD Run (known process)
  2. Calculate the derivatives (gradients)
  3. Sensitivity data
     - Mapping sent back by derivatives
  4. Update the shape (Mesh)
     - Based on the sensitivity data
     - Based on the environment constraints

- This four-step process can be run multiple times to reach an optimum evolution for the design…



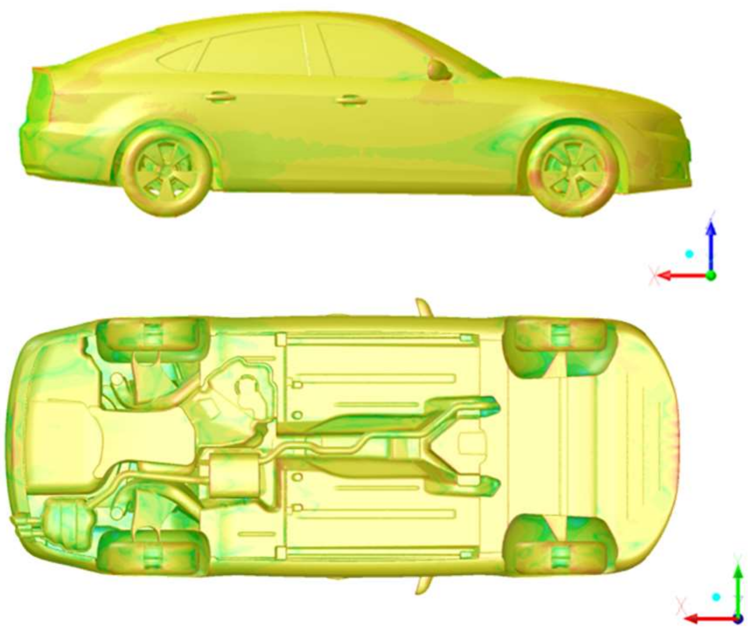
Adjoint-Driven Optimization



Additional Capabilities

- But the adjoint can do much more than just shape optimization…

Insight	Identification of the most important factors affecting performance.
Design Exploration	How a prescribed change will alter the performance.
Optimization	Systematic improvement of performance using gradient information.
Robust Design	Comprehensive identification of the most influential design parameters.
Robust Simulation	Sensitivity of the numerics to the mesh node locations.



$\log_{10}$(magnitude of shape sensitivity)

Adjoint Theory Overview

How Does the Adjoint Solver Work?

- Macro view of a *conventional flow* solved with Fluent

Inputs (c)

- Boundary mesh
- Interior mesh
- Material properties
- Boundary conditions
  - Flow angle
  - Inlet velocity
  - …
- …

Flow solver

Flow solution (q)

- Field data
  - Pressure
  - Velocity
  - Density
  - …
- Outputs
  - Contour plots
  - Vector plots
  - xy plots
  - …

Evaluation of solution

Observable (J)

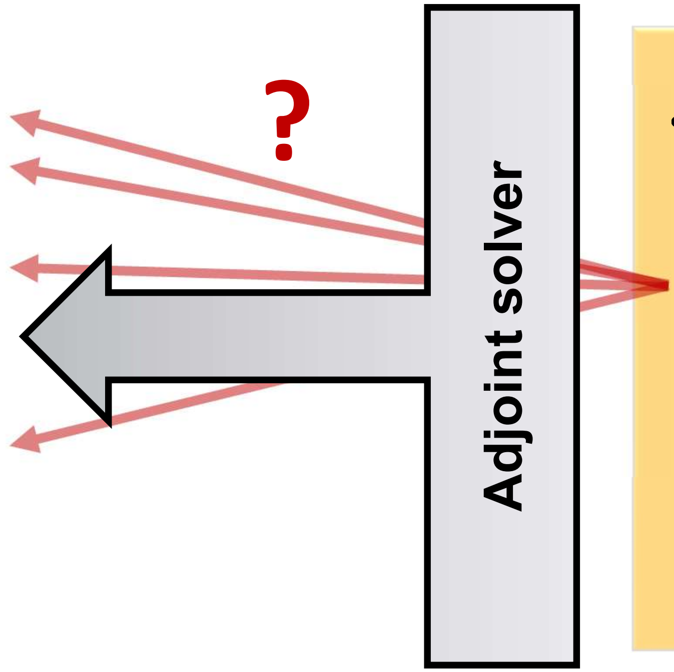
- Scalar quantities of interest
  - Lift
  - Drag
  - Pressure drop
  - …

The Adjoint Problem

- How do observables depend on the inputs?

Inputs (c)

- Boundary mesh
- Interior mesh
- Material properties
- Boundary conditions
  - Flow angle
  - Inlet velocity
  - …
- …



Observable (J)

- Scalar quantities of interest
  - Lift
  - Drag
  - Pressure drop
  - …

Mathematical Background

- We begin with a flow solution, $\underline{q}$, and the inputs to the problem, $\underline{c}$.

Quantity of interest $\boldsymbol{J}\left(\underline{\boldsymbol{q}}(\underline{\boldsymbol{c}});\underline{\boldsymbol{c}}\right)$

Residuals of the Navier-Stokes equations $\boldsymbol{R_i}\left(\underline{\boldsymbol{q}}(\underline{\boldsymbol{c}});\underline{\boldsymbol{c}}\right)=\boldsymbol{0}$

- Define the *Lagrangian* L with the vector of *Lagrange multipliers* : $\underline{\tilde{q}}^T$

Adjoint solution variables

$$\frac{dJ}{d\underline{c}}=\frac{dq}{d\underline{c}}\left(\frac{\partial J}{\partial \underline{q}}+\underline{\tilde{q}}^T\frac{\partial R}{\partial \underline{q}}\right)+\frac{\partial J}{\partial \underline{c}}+\underline{\tilde{q}}^T\frac{\partial R}{\partial \underline{c}}$$

Choose $\underline{\tilde{q}}$ such that... $\frac{\partial J}{\partial \underline{q}}+\underline{\tilde{q}}^T\frac{\partial R}{\partial \underline{q}}=0 \Rightarrow \boxed{\left[\frac{\partial R}{\partial \underline{q}}\right]^T\underline{\tilde{q}}=-\left[\frac{\partial J}{\partial \underline{q}}\right]^T}$

$$\frac{dJ}{d\underline{c}}=\frac{\partial J}{\partial \underline{c}}+\underline{\tilde{q}}^T\frac{\partial R}{\partial \underline{c}}$$

Adjoint sensitivities

☺ Reduced to one linear problem!!!!

Adjoint Sensitivities

Let's examine the Adjoint sensitivities in more detail. The sensitivity equation below is evaluated at each mesh node in our CFD model. For shape sensitivity, we consider the input vector $\underline{c}$ to be the (x,y,z) locations for every node in our model i.e. the mesh.

$$\frac{dJ}{d\underline{c}} = \frac{\partial J}{\partial \underline{c}} + \underline{\tilde{q}}^T \frac{\partial R}{\partial \underline{c}}$$

NOTE: $\frac{\partial J}{\partial \underline{c}}$ and $\frac{\partial R}{\partial \underline{c}}$ are calculated using expressions derived from the definitions of the observables and the Fluent CFD discretized equations.

Total sensitivity of J w.r.t. x,y,z at a given mesh node.

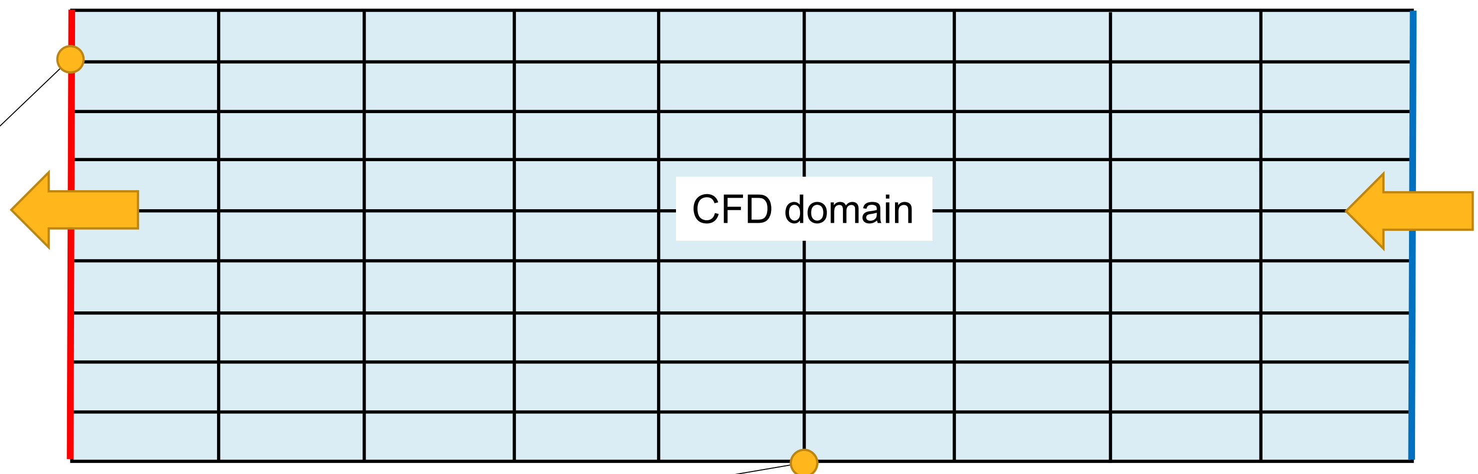
Change in J due to a change in the node's x,y,z position at a given mesh node.

Change in J due to the sensitivity of the flow solution to changes in x,y,z at a given node location. This depends on the Adjoint solution!

Adjoint Sensitivity Example

$J = \boldsymbol{mass\ flow\ at\ outlet} = \rho AV|outlet$

$c = mesh\ (x, y, z)$



Here, the observable depends directly on the geometry at the outlet node…

$\frac{\partial J}{\partial \underline{c}}$ → non-zero

$\underline{\tilde{q}}^T \frac{\partial R}{\partial \underline{c}}$ → non-zero

Here, the observable does not depend directly on geometry at the wall node…

$\frac{\partial J}{\partial \underline{c}} = 0$

$\underline{\tilde{q}}^T \frac{\partial R}{\partial \underline{c}}$ → non-zero

Mathematical Background: Summary

- Flow solution $\underline{q}$, and the inputs to the problem $\underline{c}$.

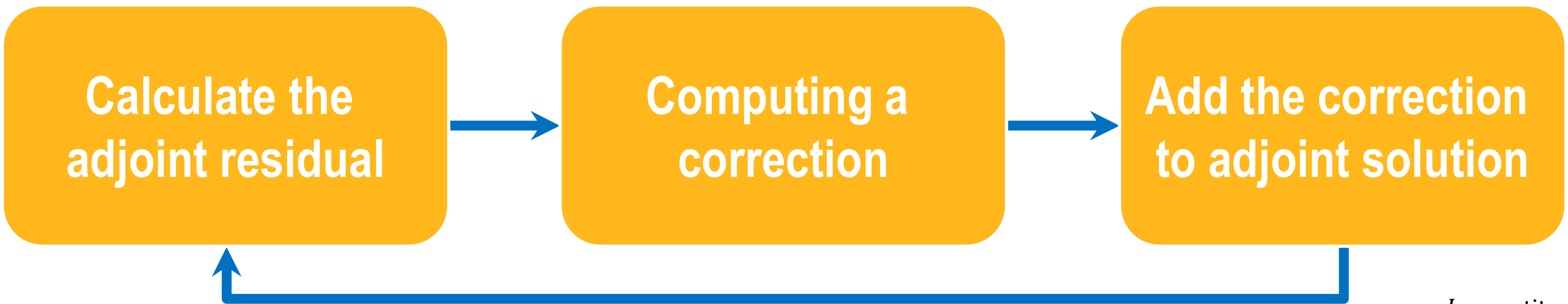
Quantity of interest $\boldsymbol{J}\left(\underline{\boldsymbol{q}}(\underline{\boldsymbol{c}});\underline{\boldsymbol{c}}\right)$

Residuals of the Navier-Stokes equations $\boldsymbol{R_i}\left(\underline{\boldsymbol{q}}(\underline{\boldsymbol{c}});\underline{\boldsymbol{c}}\right)=\mathbf{0}$

- In large-scale optimization we need the derivative (or sensitivity): $\frac{dJ}{d\underline{c}}$
- That is given by the equation: $\frac{dJ}{d\underline{c}}=\frac{\partial J}{\partial\underline{c}}+\underline{\tilde{q}}^T\frac{\partial R}{\partial\underline{c}}$
  - where $\underline{\tilde{q}}$ is solution of $\left[\frac{\partial R}{\partial\underline{q}}\right]^T\underline{\tilde{q}}=-\left[\frac{\partial J}{\partial\underline{q}}\right]^T$ (the so-called adjoint problem)

➲ The adjoint method is a very clever mathematical approach that makes possible the computation of the shape derivative in large-scale optimization

Adjoint Solver Methodology



J: quantity of interest
q: flow solution
R: Residuals of Navier-Stokes
$\tilde{R}$: Residuals of adjoint
M: simplified system Jacobian
α: URF

1. Calculate the adjoint residual: $\tilde{R} = \frac{\partial J}{\partial q_j} - \frac{\partial R_i}{\partial q_j}\tilde{q}_i$
2. Determine correction to adjoint solution: $M\Delta\tilde{q}_i = \frac{\partial J}{\partial q_j} - \frac{\partial R_i}{\partial q_j}\tilde{q}_i = \tilde{R}$
   - AMG iterative approach is used to compute an approximate solution to the equations
3. Update adjoint equations by the corrected adjoint vector: $\tilde{q}_i \leftarrow \tilde{q}_i + \alpha_i \Delta\tilde{q}_i$
4. Start again until residuals reach the specified threshold or max iterations reached

Continuous Adjoint vs Discrete Adjoint

- **Continuous adjoint**

1. Derive the adjoint equation formulation first
2. Discretize the adjoint equation.

- $D\left[\left(\frac{\partial R}{\partial q}\right)^T\right]\lambda = D[\left(\frac{\partial J}{\partial q}\right)^T]$

- **Advantages**

- Better efficiency in speed and memory

- **Discrete adjoint**

1. Have the discretization of the flow solver first
2. Get the exact discretized adjoint equation for the discretized flow solver system

$$\left[D\left(\frac{\partial R}{\partial q}\right)\right]^T \lambda = \left[D\left(\frac{\partial J}{\partial q}\right)\right]^T$$

- **Advantages**

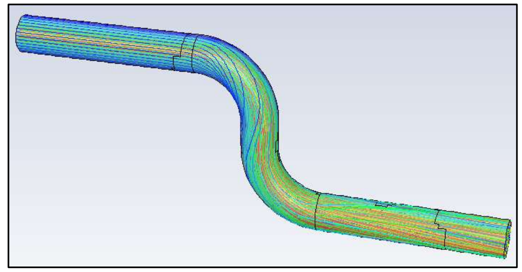
- High accuracy: discrete adjoint solver gives exact derivative of the numerical simulation system. Much better accuracy especially for the turbulence problem.
- The differentiation can be verified with finite difference method in both unit function level and system level.
- Ansys Fluent uses the **Discrete Adjoint** approach

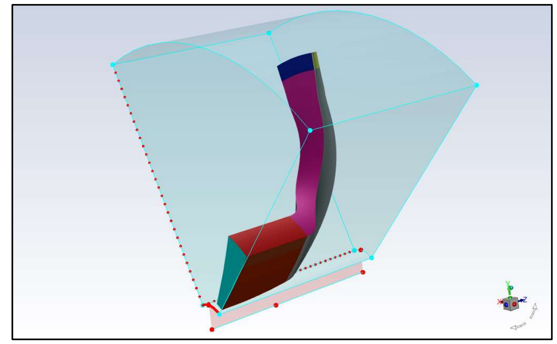
The Adjoint Solver in Ansys Fluent

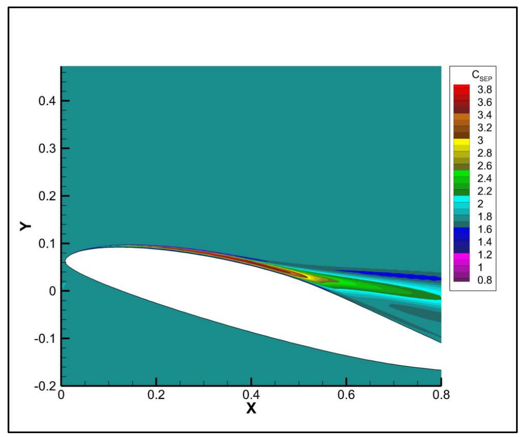
Ansys

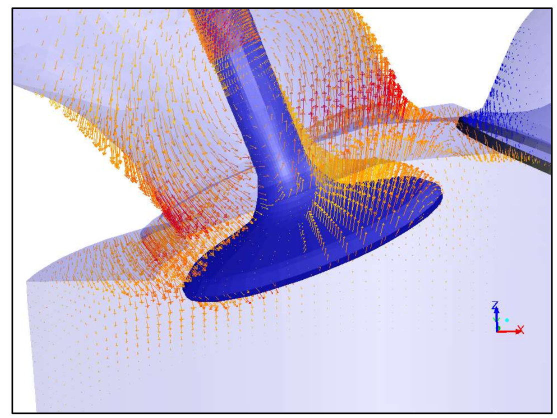
Physics Supported by the Fluent Adjoint Solver

- **Mesh**
  - All mesh types (hex, tet, wedge, polyhedral)
- **CFD Solver**
  - **Steady-state**, pressure-based solver (Segregated and Coupled)
- **Physical Models**
  - Incompressible and compressible flow
  - Energy equation
  - Laminar and Turbulent Flow (k-ε, k-ω, GEKO)
  - Moving Reference Frames
- **Materials**
  - Constant fluid/solid properties
  - Ideal gas (compressible)
- **Cell Zone Types**
  - Fluid zones, porous media
- **Boundary Conditions**
  - Walls, pressure inlets, velocity inlets, mass flow inlets, mass flow outlets, pressure outlets, pressure farfield, symmetric, periodic

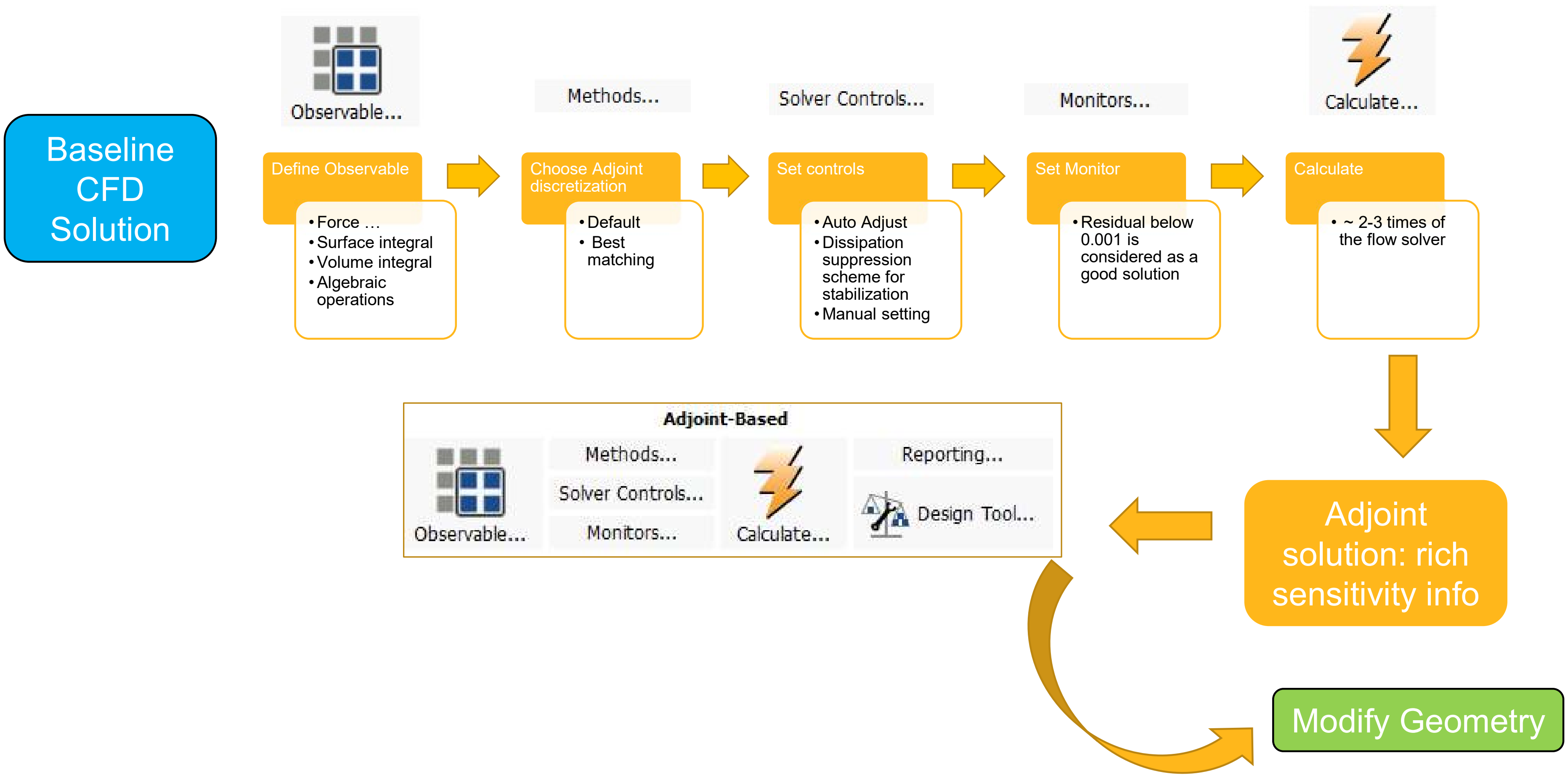






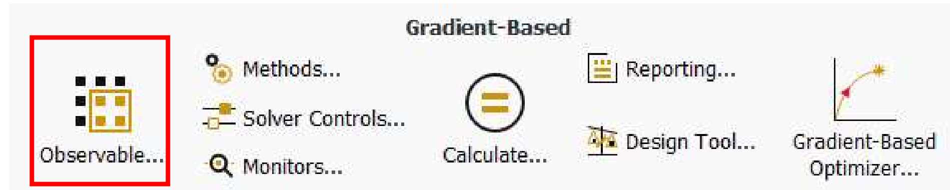


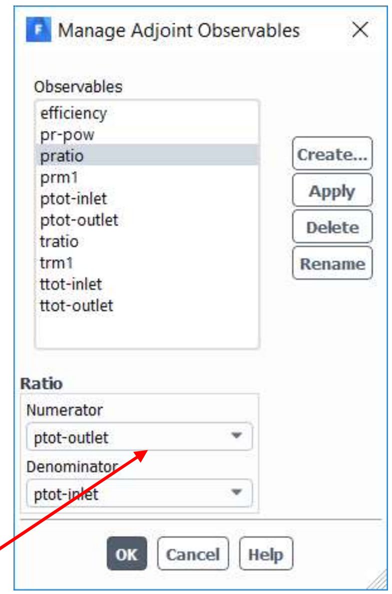
Adjoint Solver – Fluent Workflow

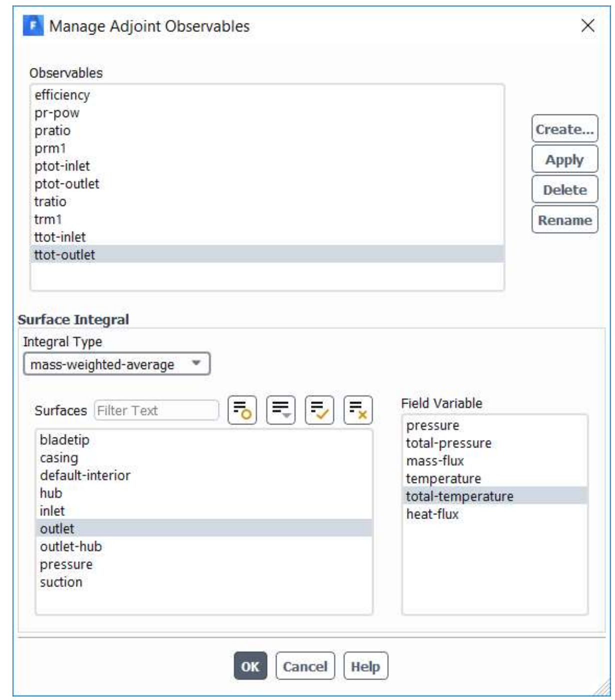


Defining Observables

- Wide range of observable quantities
  - Pressure drop (total pressure difference)
  - Forces and moments, swirl
  - Surface and volume integrals of field quantities
    - Static and total pressure
    - Static and total temperature
    - Mass flux and heat flux
  - May use Iso-clip, interior, boundary zones
- Surface and volume integrals on selected zones and cell registers
- Mathematical operators on defined observables
  - Ratio, product, linear combination, constants, unary operation, arithmetic average, mean variance
  - Operators permit you to develop custom observables



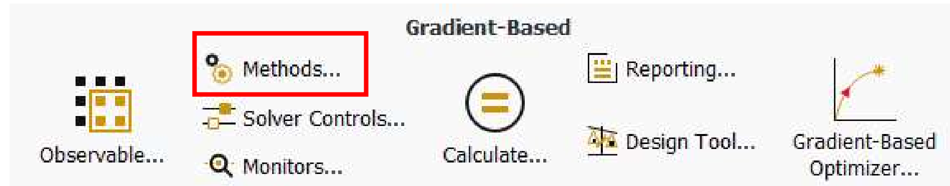


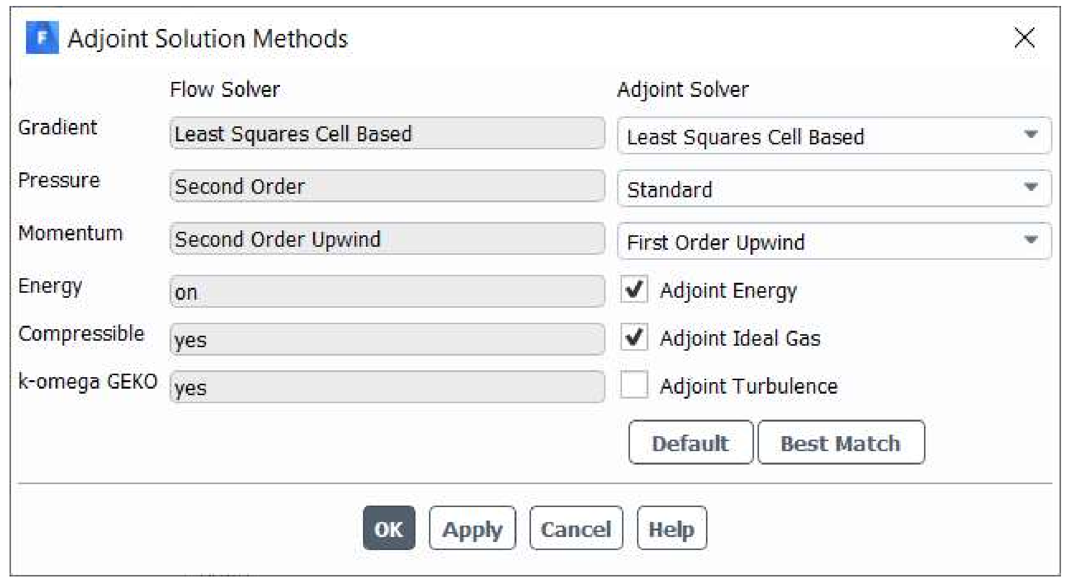


Ansys

Adjoint Solver Methods

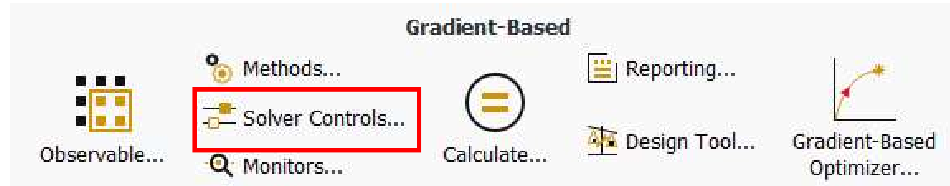
- Assign Adjoint solver methods
- Ideally, methods should match flow solution but lower order is OK and will be more stable
- Options for
  - Energy equation
  - Compressibility (Ideal Gas)
  - Adjoint Turbulence (NEW 2020R2)

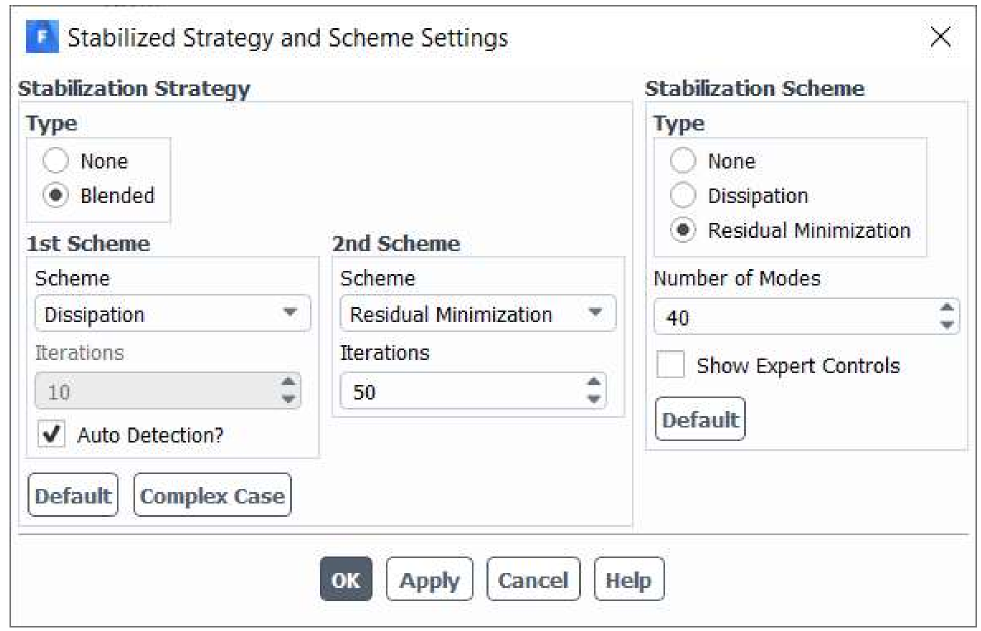


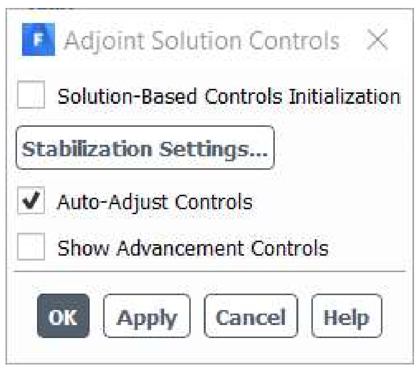


Adjoint Solver Controls

- Set up solution controls and stabilization strategy
  - Dissipation scheme
  - RMS
- More automation to assist with convergence

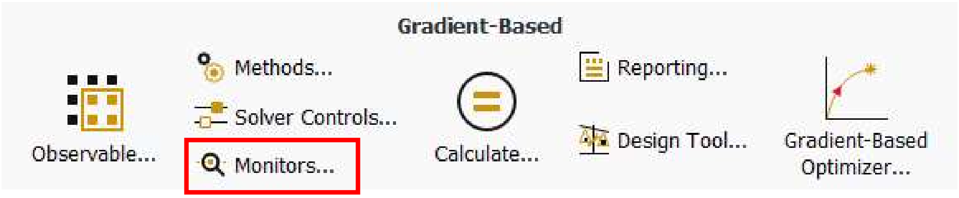


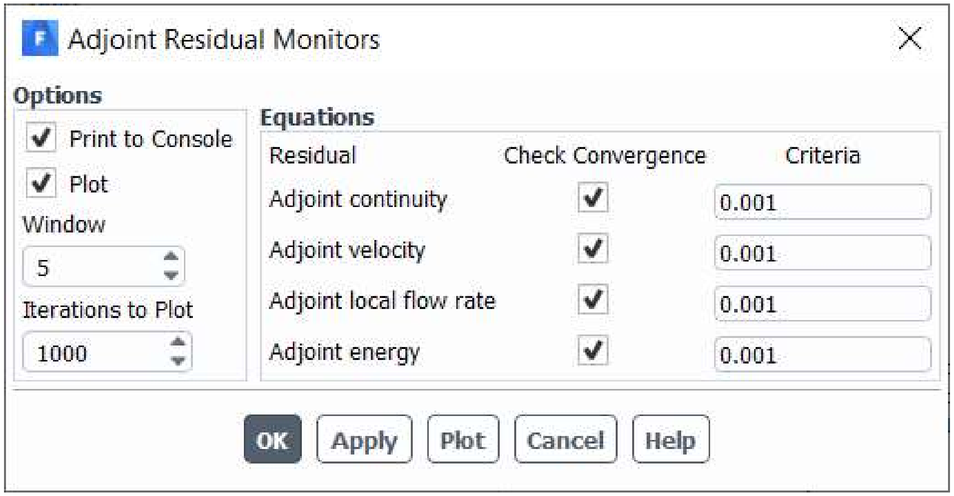




Adjoint Solver Monitors

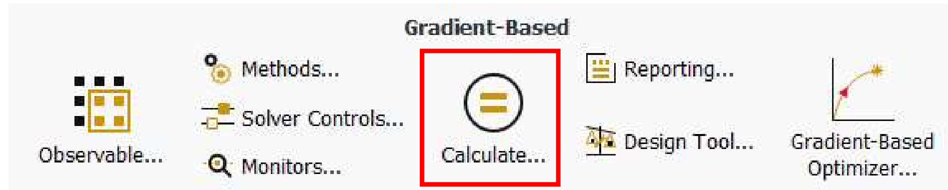
- Set Adjoint residual convergence tolerances
- Plot stored residuals

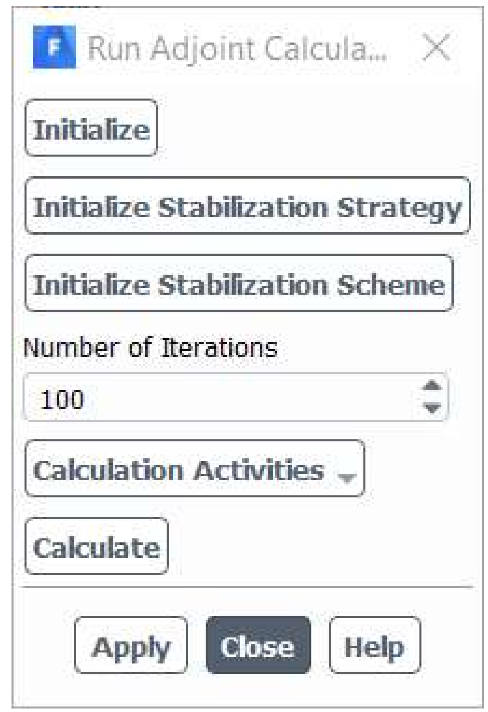




Adjoint Solver Calculation

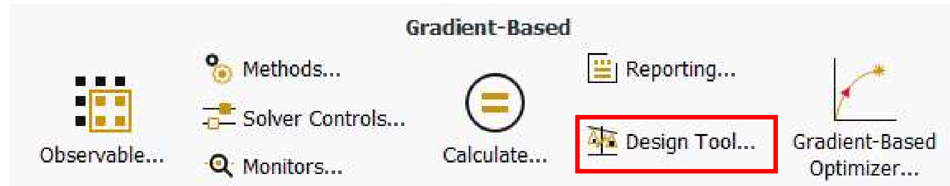
- Initialize and calculate the Adjoint solution
- Solution time is roughly equivalent to a steady-state calculation of the flow solution

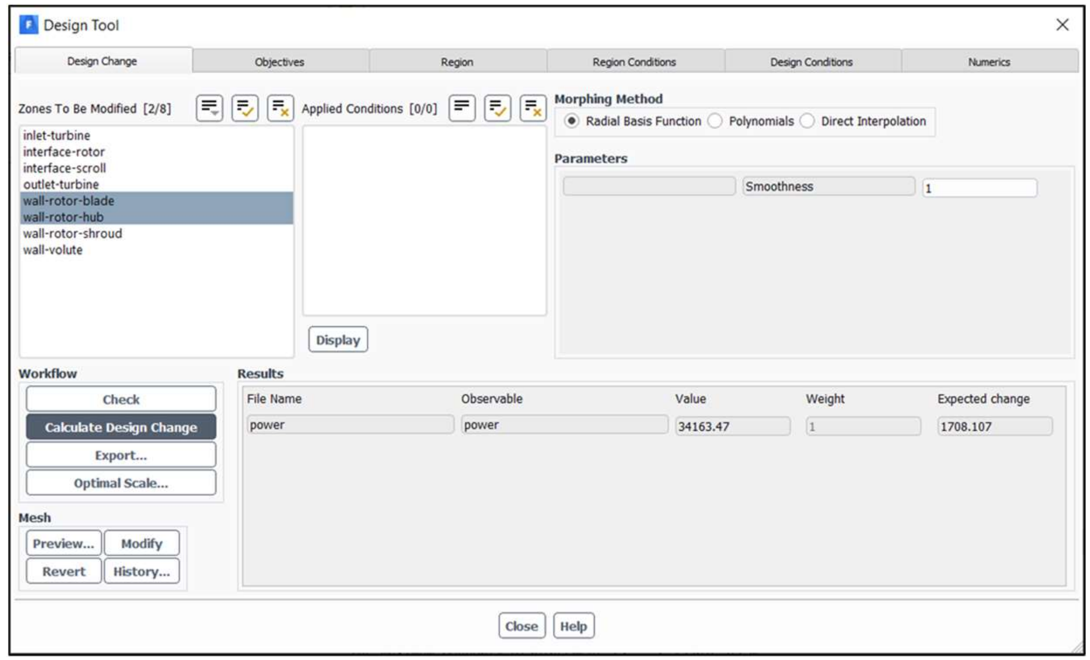




The Adjoint Design Tool

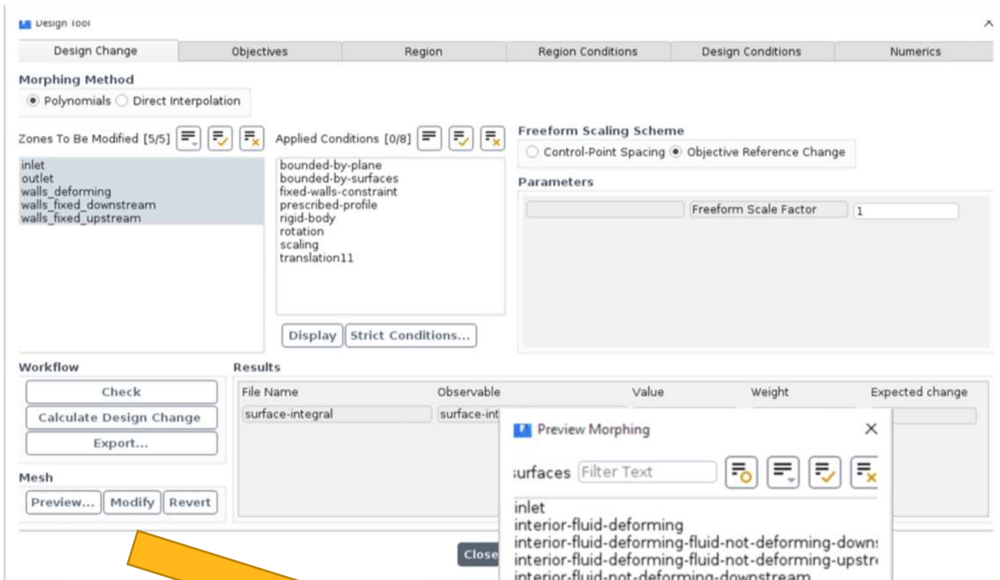
- The Design Tool allows you to set up and calculate the shape optimization design change.
  - Design change derived from computed Adjoint solution
  - New design realized by morphing the mesh according to the calculated design change
  - Controls available for zone selection, objective, design constraints, region, smoothness, numerics for design change calculation
  - Polynomial and Direct Interpolation methods for defining new surface shapes
  - Modified geometry can now be previewed before morphing! (next slide)
  - Can undo mesh morphing (Revert)
  - Can import/export the sensitivity fields

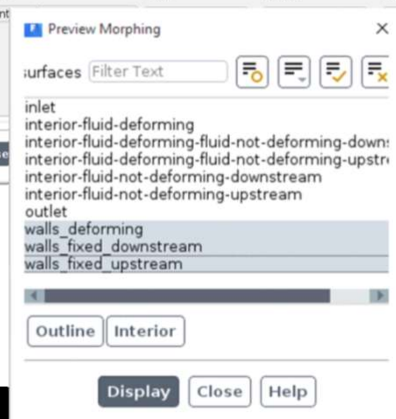


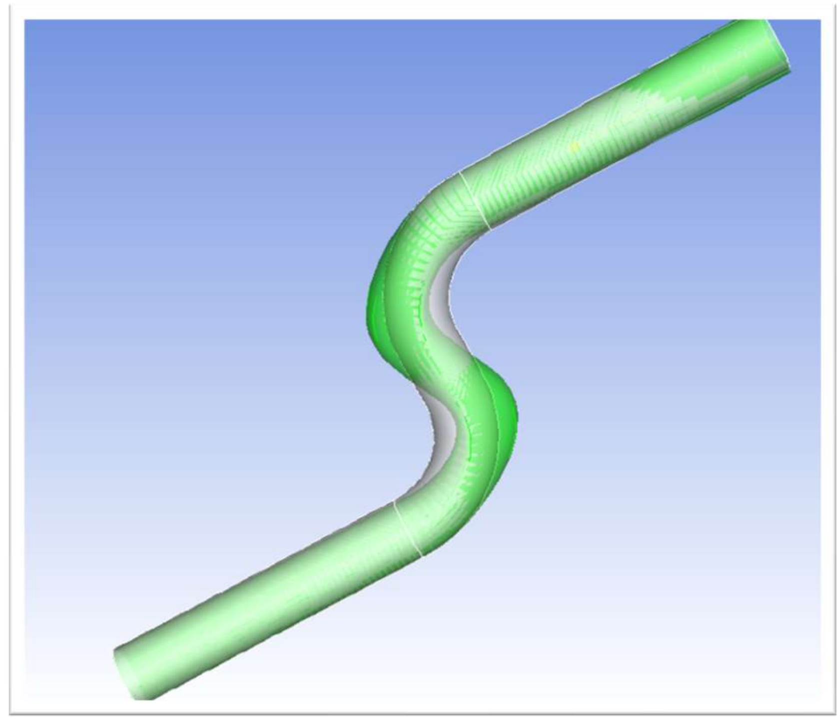


Previewing the Design Changes

The morphing can be previewed and compared with the original mesh after optimal displacement is computed.

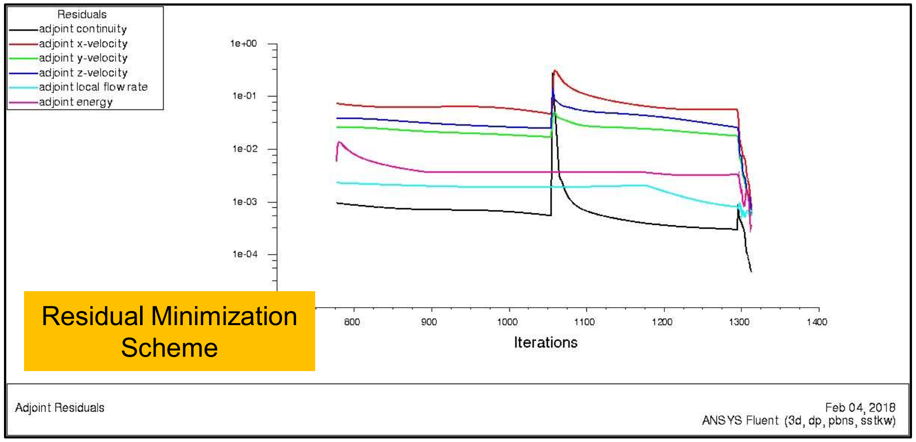


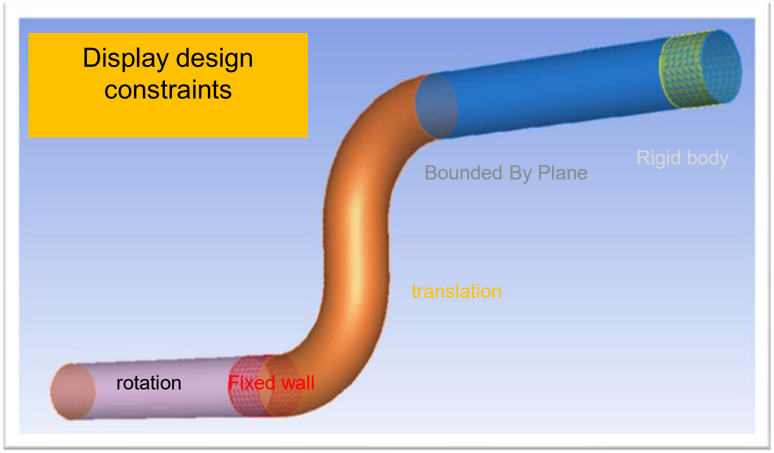




Adjoint Solver Improvements

- Overview of recent Adjoint improvements
  - More observables
  - More boundary conditions
  - Residual Minimization Scheme (RMS)
  - Direct Interpolation and Radial Basis Functions for morphing (complements Polynomial Method)
  - Design Tool Usability (refined interface, new options)
  - Support for GEKO turbulence model in Adjoint calculation
    - k and omega are included in observable selections
    - More details in the Appendix





Adjoint Output – Shape Sensitivities and Normal Displacement

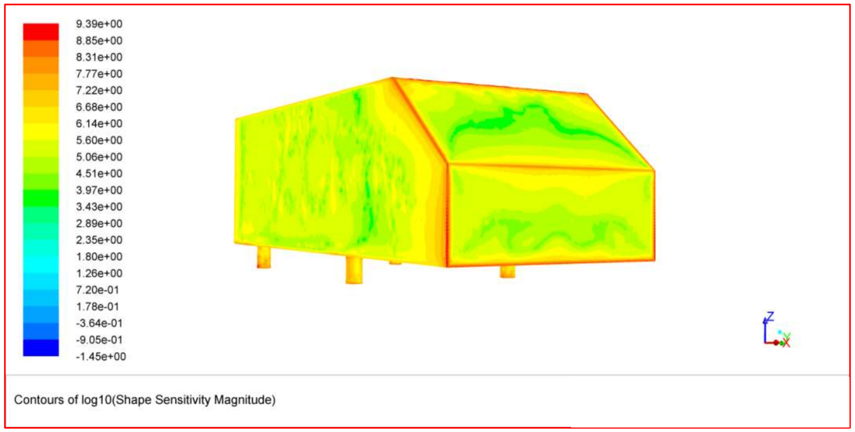
Log10(Shape Sensitivity Magnitude)

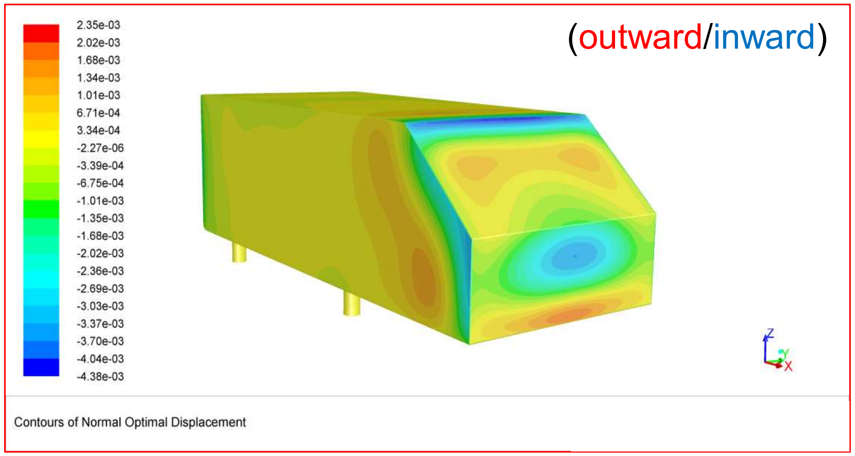
- Display the regions with highest sensitivity to the observable.
- Areas with large values show increased impact on the observable if the mesh is changed locally.

Normal Optimal Displacement

- Shows the displacement of the geometry to reach calculated design change.
- Normal optimal displacement values are in the used unit for length in Fluent.
- Positive values mean a design change towards the volume mesh and vice versa for negative values.
- Use normal optimal displacement plot to check the proposed design change.

There are many other sensitivity variables that can be plotted

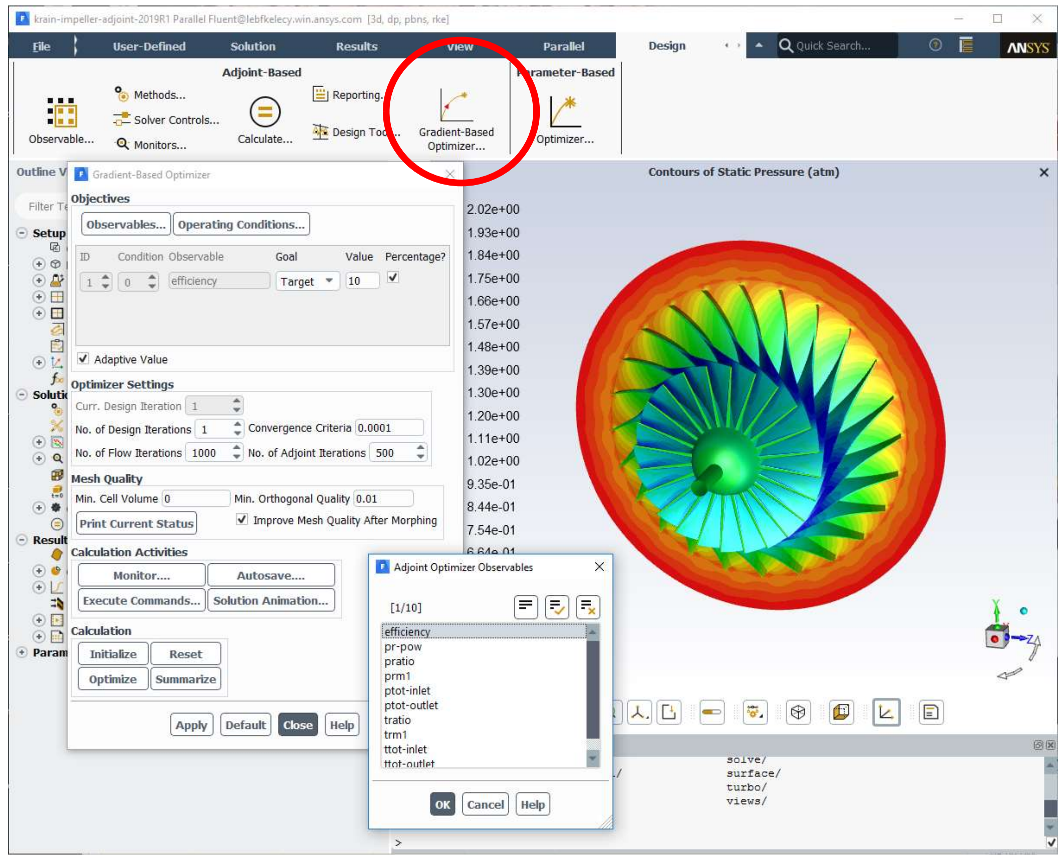




Gradient-Based Optimizer

Workflow Automation – The Gradient-Based Optimizer

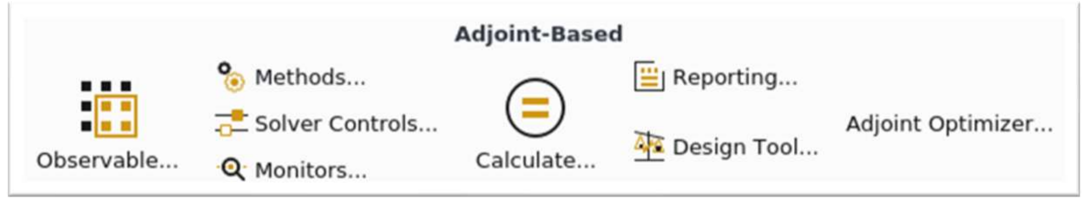
- The Gradient-Based Optimizer provides a new level of workflow automation to the Adjoint solver.
- Permits both multiple objective and multiple flow condition optimization
  - For example, you can perform a shape optimization on an airfoil which attempts to increase lift and reduce drag over a range of flow BCs.
- You can also use this interface to automate multiple design points for single objective cases (an example will be shown).
- Additional features for mesh quality preservation, design point tracking, file autosaving, animations, and more!

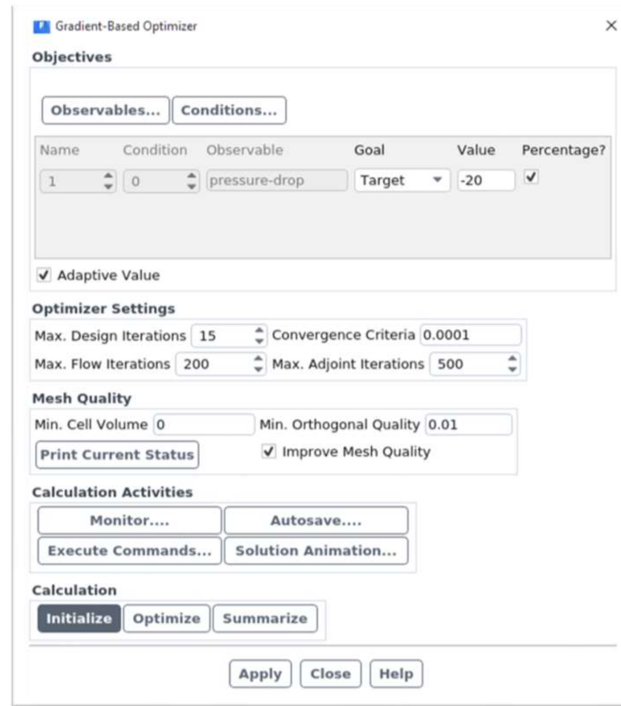


Gradient-Based Optimizer Workflow

Multi-Design Point and Multi-Objective Workflows

- Select observables/conditions of interest
- Flow & Adjoint Solvers
  - (optional) Set up Methods
  - (optional) Set up solver controls
- Design Space & Constraints
  - Morphing method & Region
  - Surfaces to be modified
  - (optional) Region condition
  - (optional) Design condition
  - (optional) Numerics
- Configure gradient-based optimizer
- Run the optimization
  - Initialize
  - Optimize

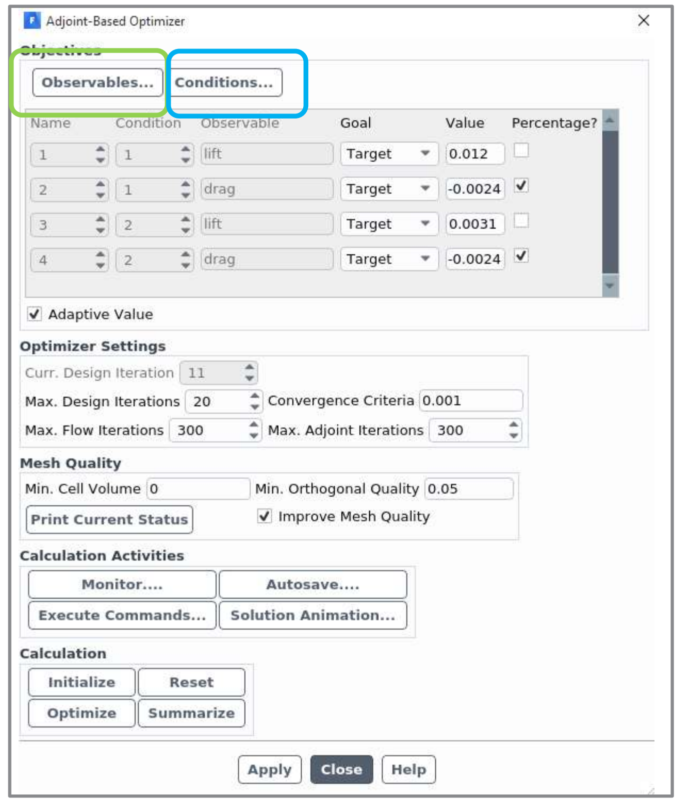


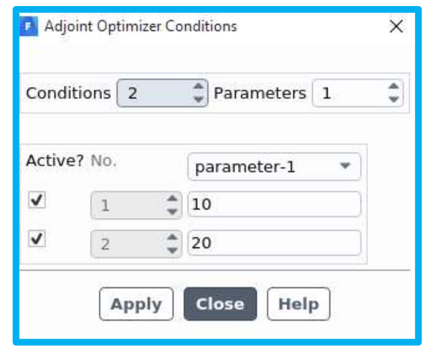


Objectives

GUIs for input of objective settings

- Use adjoint optimizer Observable panel to define multiple observables.
- Use adjoint optimizer Condition panel to define multiple operating condition using **input parameters.**
- In this example, the input parameter is the inlet velocity at the operating condition of both 10 m/s and 20 m/s. We will later show an example where we attempt to reduce the drag and increase lift on a body for two inlet velocities (10 m/s and 20 m/s) simultaneously.



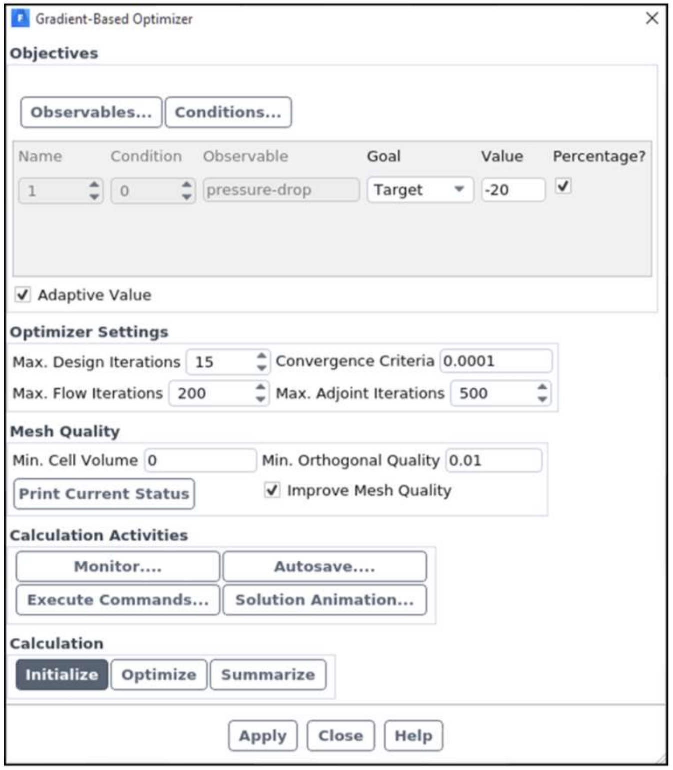


Optimizer Settings

- Goal: Make sure **the flow solver, adjoint solver, and design tool all converge**.
    - Adjoint solver setting recommendation
        - Default adjoint method
        - Default adjoint setting with blended auto-adjust
    - Design tool setting recommendation
        - Use polynomial method without design constraint and with light design constraint.
        - Use direct interpolation method with design constraints.
- Convergence Criteria:
    - **expected change/initial observable** < Convergence Criteria

      or

    - **real change/initial observable** < Convergence Criteria
- If the above condition is satisfied for **all the cases**, the optimization is considered as converged.
- The optimization generally reaches convergence in about 10-20 design iterations, depending on the problem and settings.

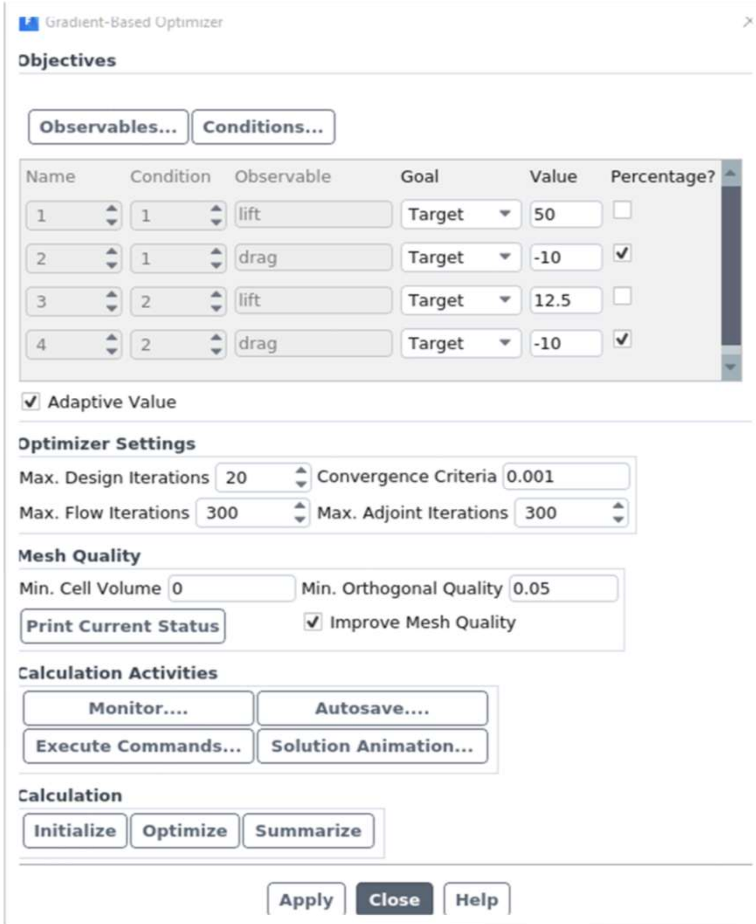


Mesh Quality

- Defines the minimum requirement for the mesh quality
  - If the mesh quality requirements are not met after a design calculation, the target change will be reduced until the mesh quality requirement is met. Then the design calculation is performed again. **This avoids issues due to negative volume after large morphing using design tool.**
  - If the mesh quality requirement is not satisfied, the optimization will stop.
  - A Min. Orthogonal Quality large than 0 is recommended to avoid left-handed faces
- Print Current Status

```
min          cell-volume: limit    0.00000e+00; current value    3.94817e-04.
min   orthogonal-quality: limit    5.00000e-02; current value    5.92320e-01.
```

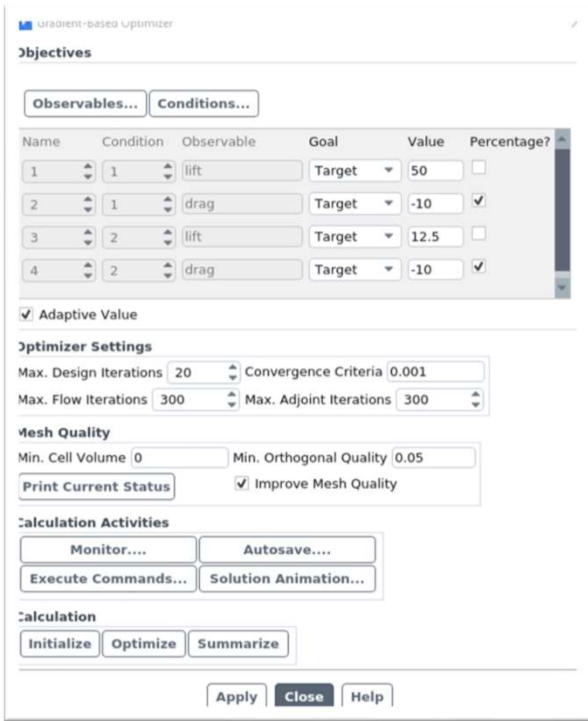
- Improve mesh quality after morphing
  - Using the tui command 3 times:
    - /mesh/repair-improve/improve-quality
  - **Note: This operation may deteriorate the mesh quality in some situations.**

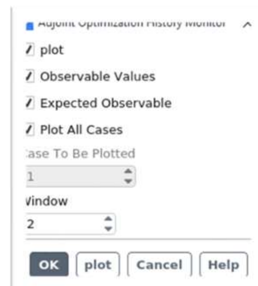


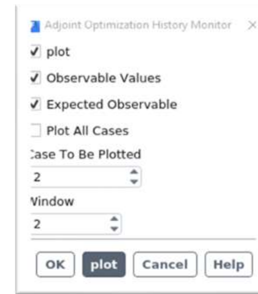
Monitoring

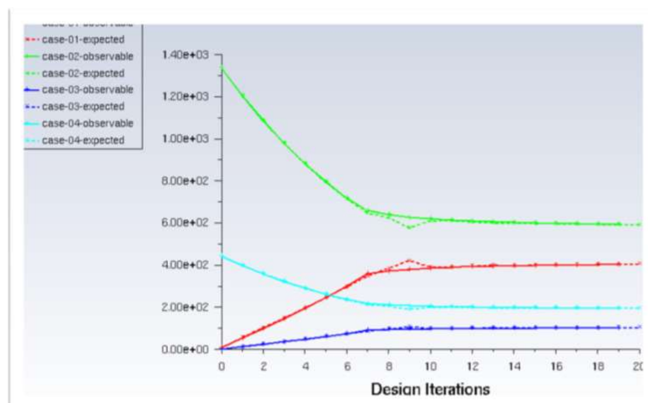
Optimization history can be monitored as the calculations proceed.

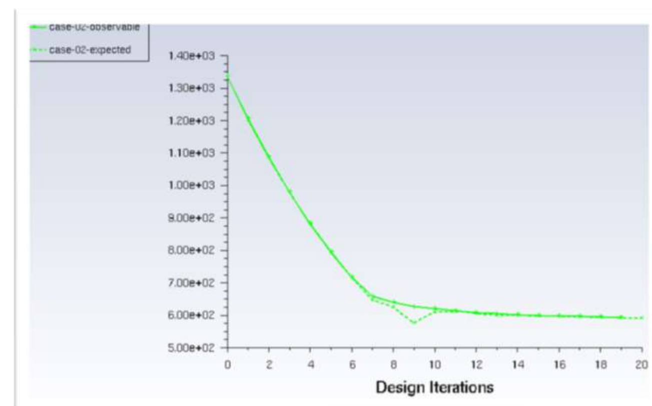
- The observable value of the case for the current design iteration, as solid lines.
- The observable value predicted by the adjoint solver for the next design iteration, as dashed lines.
- For multi-objective and multi-condition optimization, you can also plot each individual case as well







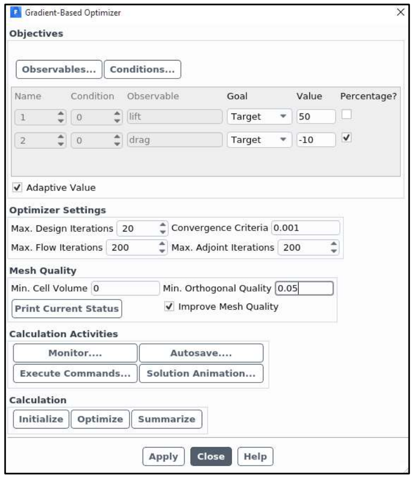


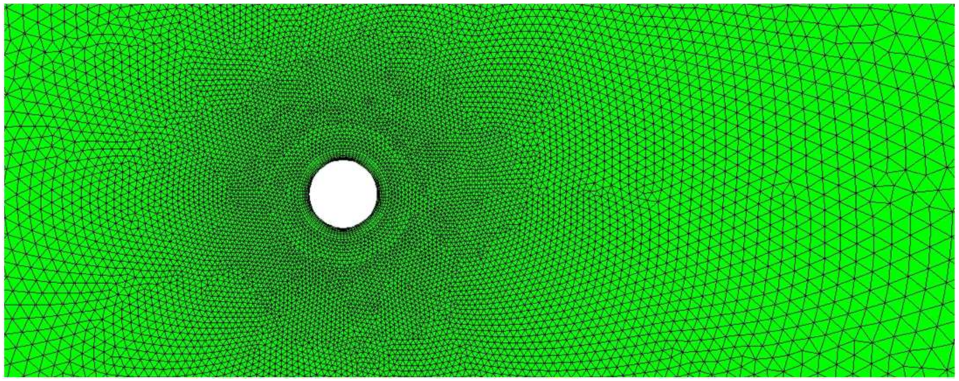


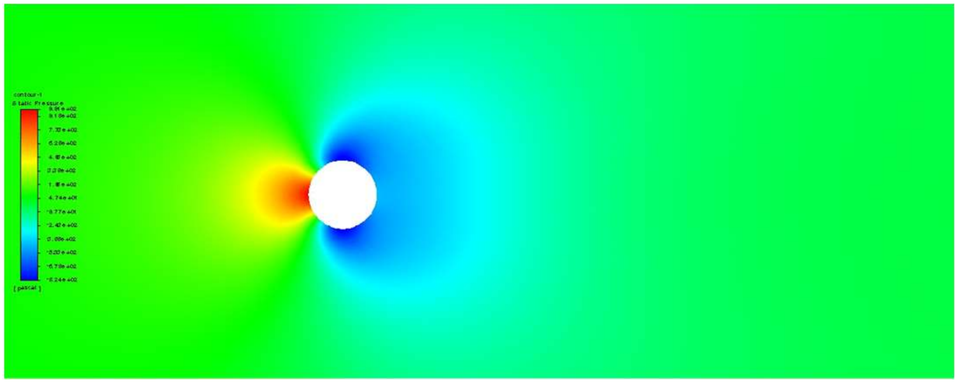
Multi-Objective Optimization Example: 2D Cylinder

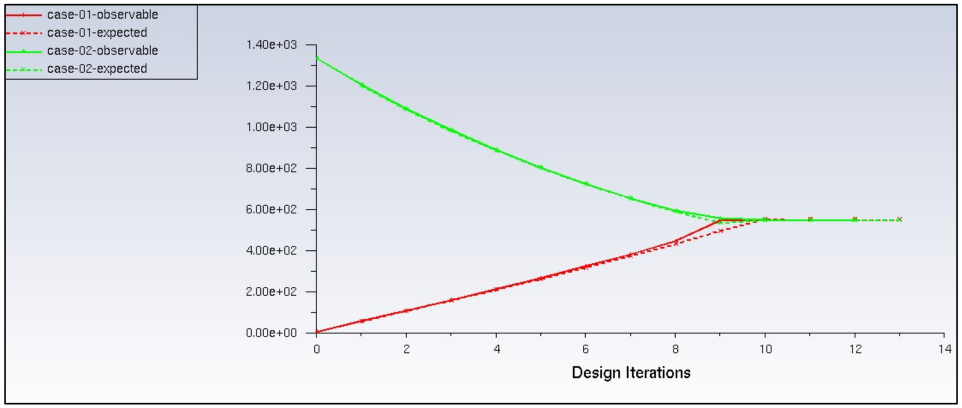
Goals:
- Increase lift
- Reduce drag
- Single flow condition

Flow:
- Inlet velocity 40m/s

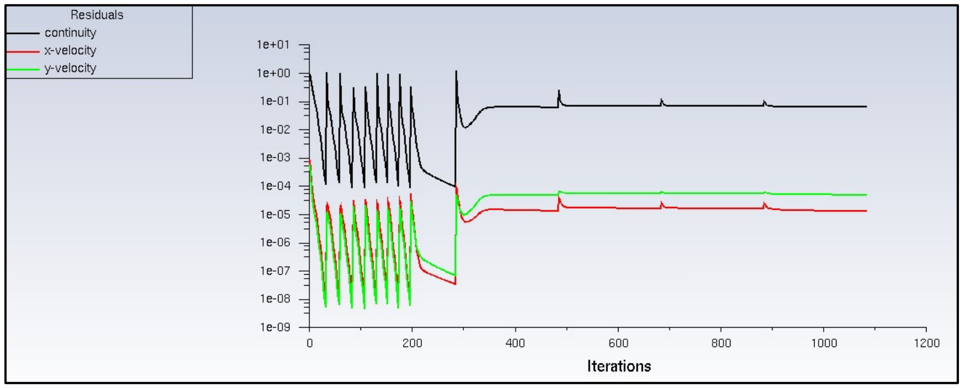


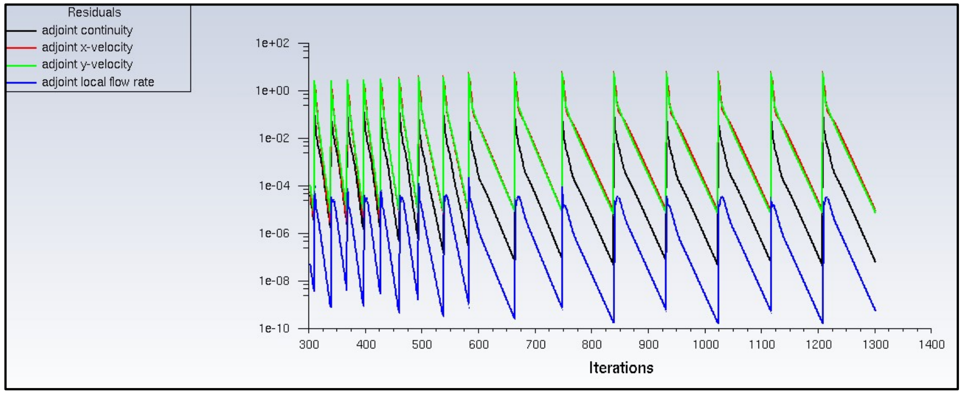






Multi-Objective Optimization Example: 2D Cylinder





1 Design iteration
=
1 flow simulation
+
2 adjoint simulations
+
1 design change

Multi-Objective Optimization Example: 2D Cylinder

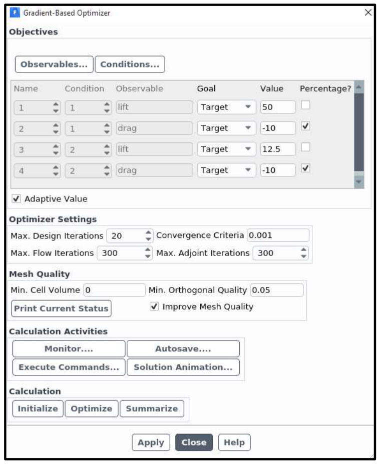
Design Iteration	Case	Condition	Observable	Flow Convergence	Adjoint Convergence	Observable Value	Expected Change
0	1	0	lift	Y	Y	8.129e+00	5.000e+01
0	2	0	drag	Y	Y	1.338e+03	-1.338e+02
1	1	0	lift	Y	Y	5.912e+01	5.000e+01
1	2	0	drag	Y	Y	1.209e+03	-1.209e+02
2	1	0	lift	Y	Y	1.102e+02	5.000e+01
2	2	0	drag	Y	Y	1.093e+03	-1.093e+02
3	1	0	lift	Y	Y	1.616e+02	5.000e+01
3	2	0	drag	Y	Y	9.884e+02	-9.884e+01
4	1	0	lift	Y	Y	2.142e+02	5.000e+01
4	2	0	drag	Y	Y	8.934e+02	-8.934e+01
5	1	0	lift	Y	Y	2.685e+02	5.000e+01
5	2	0	drag	Y	Y	8.071e+02	-8.071e+01
6	1	0	lift	Y	Y	3.256e+02	5.000e+01
6	2	0	drag	Y	Y	7.289e+02	-7.289e+01
7	1	0	lift	Y	Y	3.835e+02	5.000e+01
7	2	0	drag	Y	Y	6.582e+02	-6.582e+01
8	1	0	lift	Y	Y	4.474e+02	5.000e+01
8	2	0	drag	Y	Y	5.965e+02	-5.965e+01
9	1	0	lift	N	Y	5.491e+02	6.250e+00
9	2	0	drag	N	Y	5.585e+02	-6.982e+00
10	1	0	lift	N	Y	5.515e+02	7.812e-01
10	2	0	drag	N	Y	5.510e+02	-8.609e-01
11	1	0	lift	N	Y	5.522e+02	3.906e-01
11	2	0	drag	N	Y	5.509e+02	-4.304e-01
12	1	0	lift	N	Y	5.526e+02	3.906e-01
12	2	0	drag	N	Y	5.505e+02	-4.301e-01

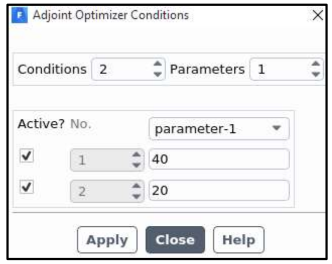
Design Iteration	min cell-volume	min orthogonal-quality
0	3.94817e-04	5.92320e-01
1	3.54433e-04	7.66153e-01
2	3.13941e-04	7.67968e-01
3	2.72518e-04	7.67043e-01
4	2.29352e-04	7.53421e-01
5	1.81829e-04	7.07991e-01
6	1.01534e-04	6.16894e-01
7	7.04313e-05	4.84953e-01
8	4.08923e-05	3.44075e-01
9	9.76970e-06	1.35852e-01
10	8.92003e-06	1.22301e-01
11	8.83860e-06	1.26644e-01
12	8.75670e-06	1.32550e-01

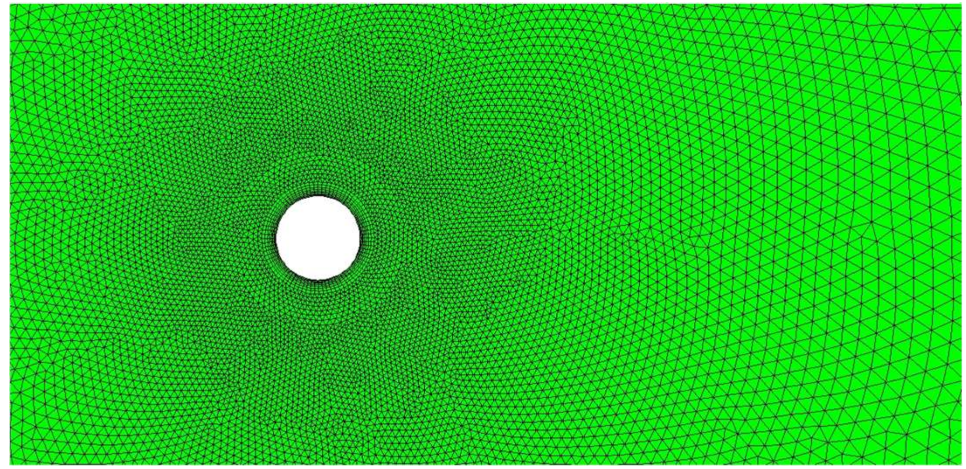
Lift: 8.12N -> 550.5 N
Drag: 1338N -> 552.6 N

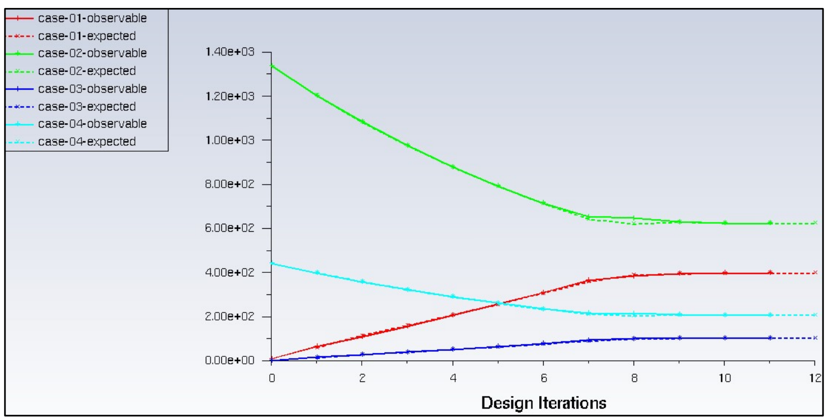
Multi-Objective, Multi-Condition Optimization Example: 2D Cylinder

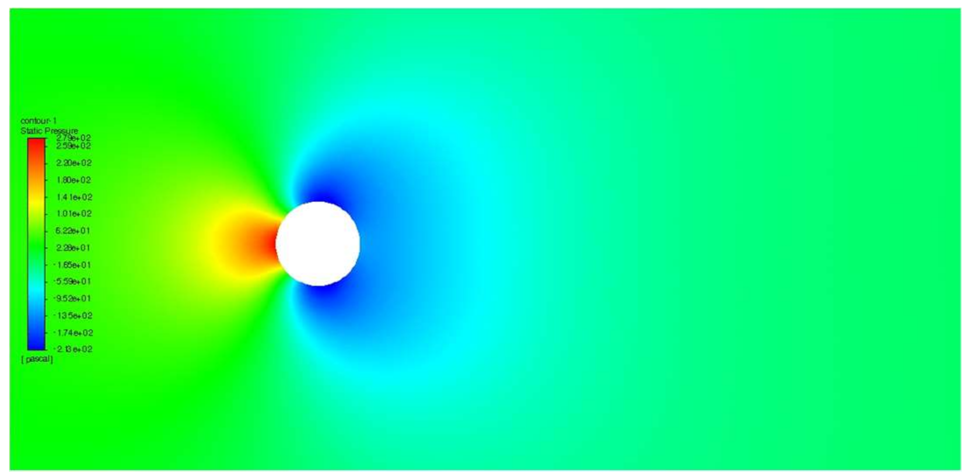
Goals:
- Increase lift
- Reduce drag
- Multiple flow conditions
- Inlet velocity set to both 20m/s and 40 m/s.



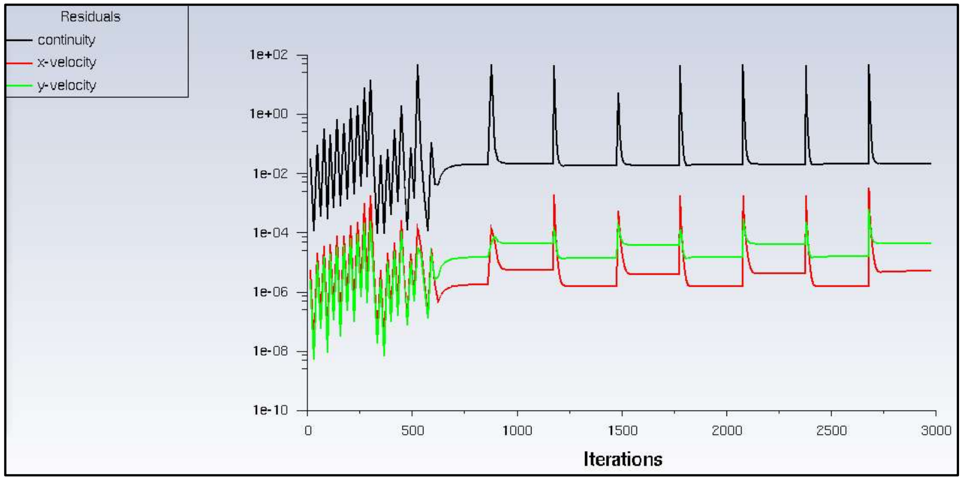


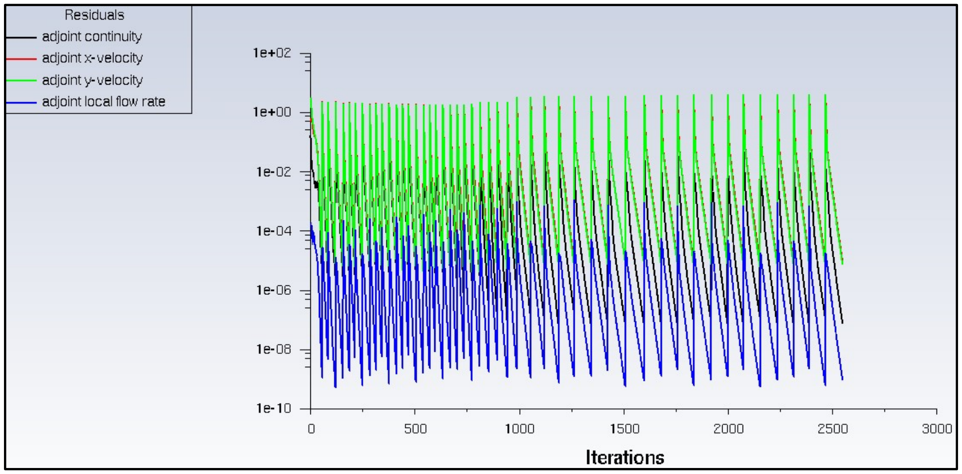






Multi-Objective, Multi-Condition Optimization Example: 2D Cylinder





1 Design iteration
=
2 flow simulation
+
4 adjoint simulations
+
1 design change

Multi-Objective, Multi-Condition Optimization Example: 2D Cylinder

Design Iteration	Case	Condition	Observable	Flow Convergence	Adjoint Convergence	Observable Value	Expected Change
0	1	1	lift	Y	Y	8.129e+00	5.000e+01
0	2	1	drag	Y	Y	1.338e+03	-1.338e+02
0	3	2	lift	Y	Y	1.076e+00	1.250e+01
0	4	2	drag	Y	Y	4.430e+02	-4.430e+01
1	1	1	lift	Y	Y	6.411e+01	5.000e+01
1	2	1	drag	Y	Y	1.203e+03	-1.203e+02
1	3	2	lift	Y	Y	1.531e+01	1.250e+01
1	4	2	drag	Y	Y	3.978e+02	-3.978e+01
2	1	1	lift	Y	Y	1.089e+02	5.000e+01
2	2	1	drag	Y	Y	1.085e+03	-1.085e+02
2	3	2	lift	Y	Y	2.673e+01	1.250e+01
2	4	2	drag	Y	Y	3.583e+02	-3.583e+01
3	1	1	lift	Y	Y	1.561e+02	5.000e+01
3	2	1	drag	Y	Y	9.770e+02	-9.770e+01
3	3	2	lift	Y	Y	3.865e+01	1.250e+01
3	4	2	drag	Y	Y	3.223e+02	-3.223e+01
4	1	1	lift	Y	Y	2.057e+02	5.000e+01
4	2	1	drag	Y	Y	8.798e+02	-8.798e+01
4	3	2	lift	Y	Y	5.110e+01	1.250e+01
4	4	2	drag	Y	Y	2.900e+02	-2.900e+01
5	1	1	lift	Y	Y	2.560e+02	5.000e+01
5	2	1	drag	Y	Y	7.927e+02	-7.927e+01
5	3	2	lift	Y	Y	6.393e+01	1.250e+01
5	4	2	drag	Y	Y	2.612e+02	-2.612e+01
6	1	1	lift	Y	Y	3.083e+02	5.000e+01
6	2	1	drag	Y	Y	7.147e+02	-7.147e+01
6	3	2	lift	Y	Y	7.742e+01	1.250e+01
6	4	2	drag	Y	Y	2.356e+02	-2.356e+01
7	1	1	lift	Y	Y	3.650e+02	2.500e+01
7	2	1	drag	Y	Y	6.532e+02	-3.266e+01
7	3	2	lift	Y	Y	9.344e+01	6.250e+00
7	4	2	drag	Y	Y	2.158e+02	-1.079e+01
8	1	1	lift	N	Y	3.825e+02	1.250e+01
8	2	1	drag	N	Y	6.456e+02	-1.614e+01
8	3	2	lift	N	Y	9.946e+01	3.125e+00
8	4	2	drag	N	Y	2.141e+02	-5.352e+00
9	1	1	lift	N	Y	3.943e+02	3.125e+00
9	2	1	drag	N	Y	6.289e+02	-3.931e+00
9	3	2	lift	N	Y	1.027e+02	7.812e-01
9	4	2	drag	N	Y	2.089e+02	-1.305e+00
10	1	1	lift	N	Y	3.981e+02	7.812e-01
10	2	1	drag	N	Y	6.249e+02	-9.764e-01
10	3	2	lift	N	Y	1.037e+02	1.953e-01
10	4	2	drag	N	Y	2.076e+02	-3.244e-01
11	1	1	lift	N	Y	3.986e+02	3.906e-01
11	2	1	drag	N	Y	6.237e+02	-4.873e-01
11	3	2	lift	N	Y	1.038e+02	9.766e-02
11	4	2	drag	N	Y	2.072e+02	-1.619e-01

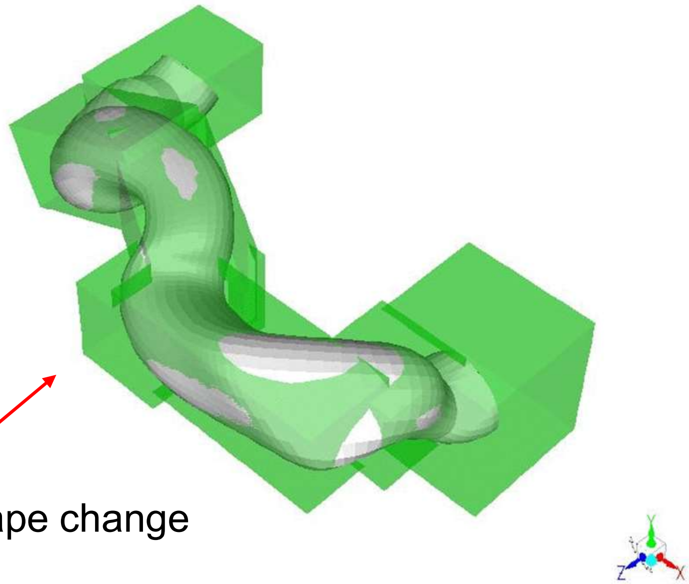
Design Iteration	min cell-volume	min orthogonal-quality
0	3.94817e-04	5.92320e-01
1	2.91771e-04	7.66287e-01
2	2.11120e-04	7.41764e-01
3	1.49376e-04	6.89075e-01
4	1.02696e-04	6.18936e-01
5	6.34678e-05	5.33071e-01
6	3.28072e-05	3.53719e-01
7	7.93379e-06	1.84503e-01
8	2.42801e-06	1.36433e-01
9	2.48062e-06	1.52513e-01
10	2.45605e-06	1.58533e-01
11	2.44840e-06	1.63234e-01

Inlet Velocity 20 m/s:
Lift: 1.07N -> 103.8 N
Drag: 443N -> 207 N

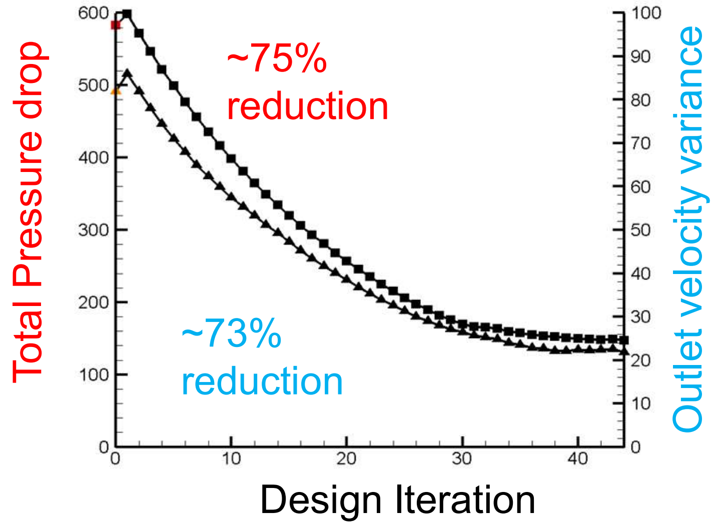
Inlet Velocity 40 m/s:
Lift: 8.12N -> 398.6 N
Drag: 1338N -> 623.7 N

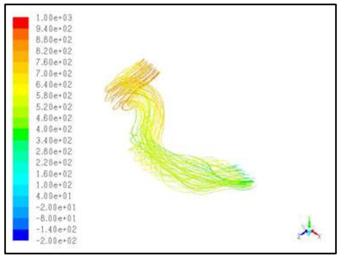
Shape Optimization Examples

Complex Duct Shape (Single Inlet and Outlet)

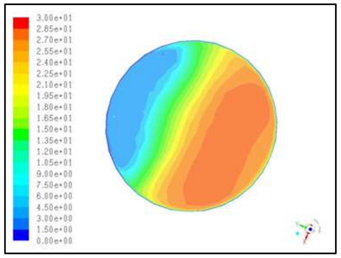


Organic surface shape change





Total pressure



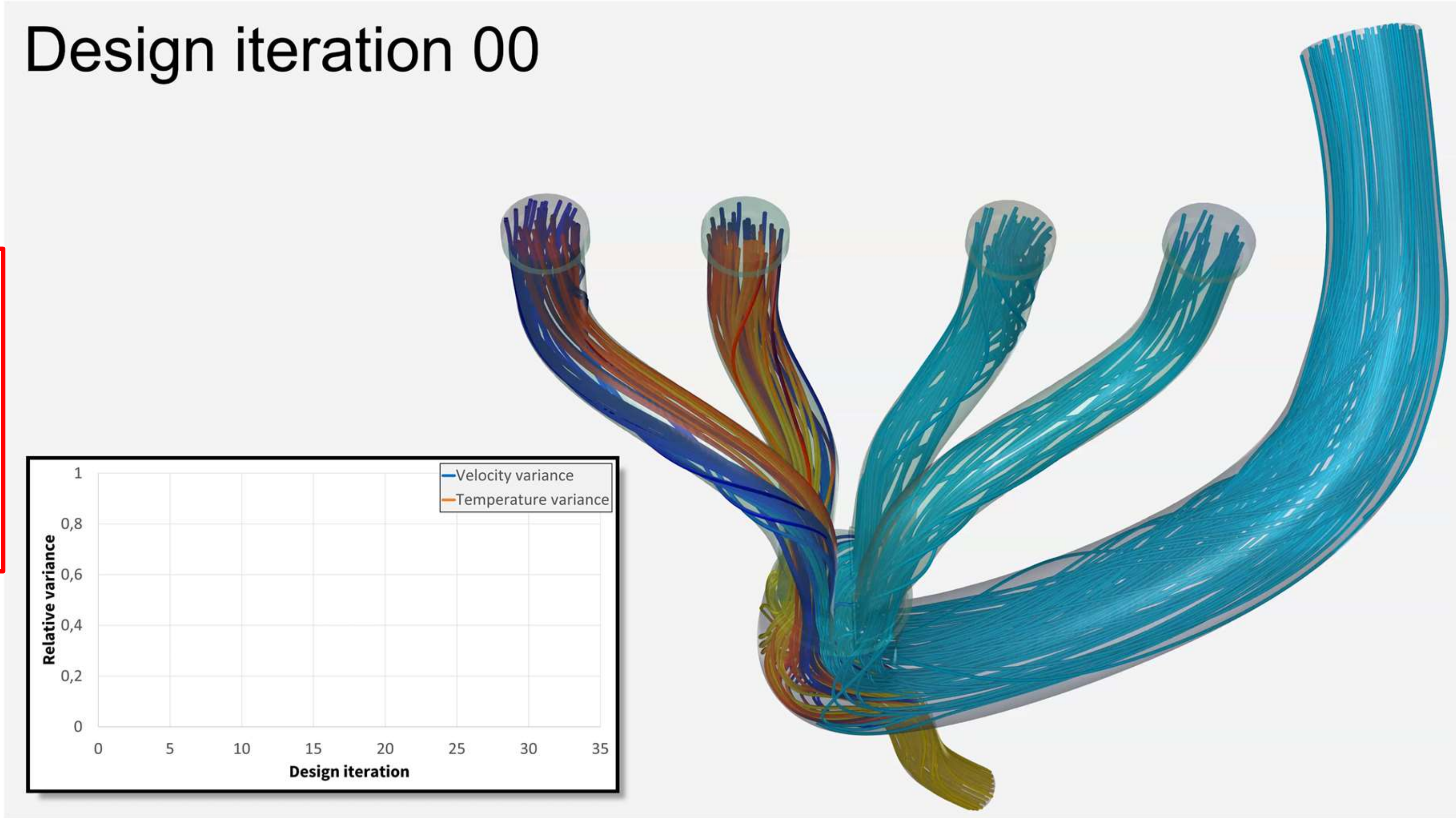
Outflow velocity profile

Goals: Reduce Total Pressure Drop and Increase Flow Uniformity

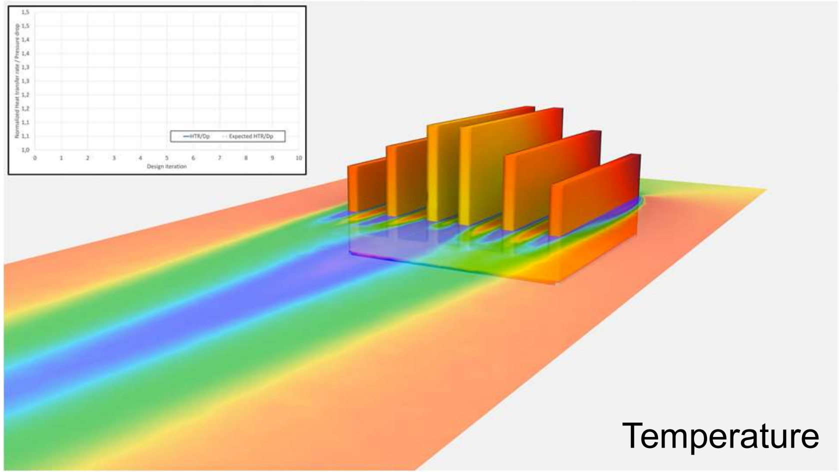
Manifold Flow Optimization

Goal: Reduce flow variance at manifold outlets

- **30** design iterations
- Velocity variance is **reduced by 35%**
- Temperature variance is **reduced by 80%.**

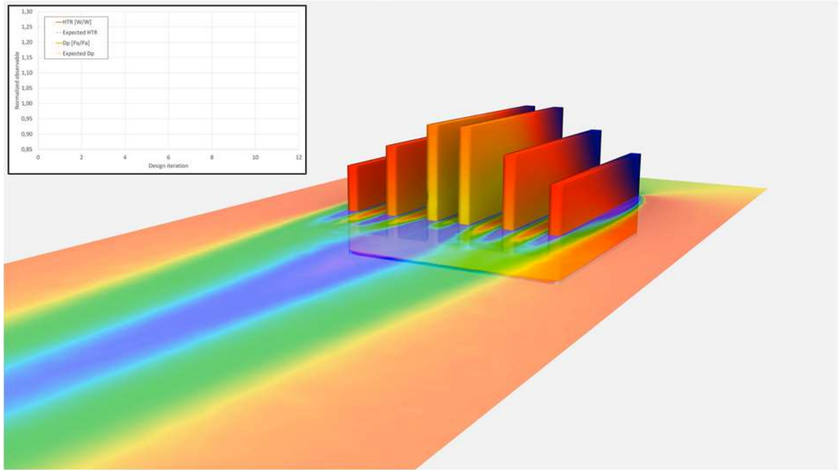


Heat Exchanger Optimization



Single objective optimization:

- Goal: increase **HTR/Δp (HTR: heat flux rate, Δp: pressure drop)**
- Result: with **9** design iterations, the ratio is **increased by 40%.**



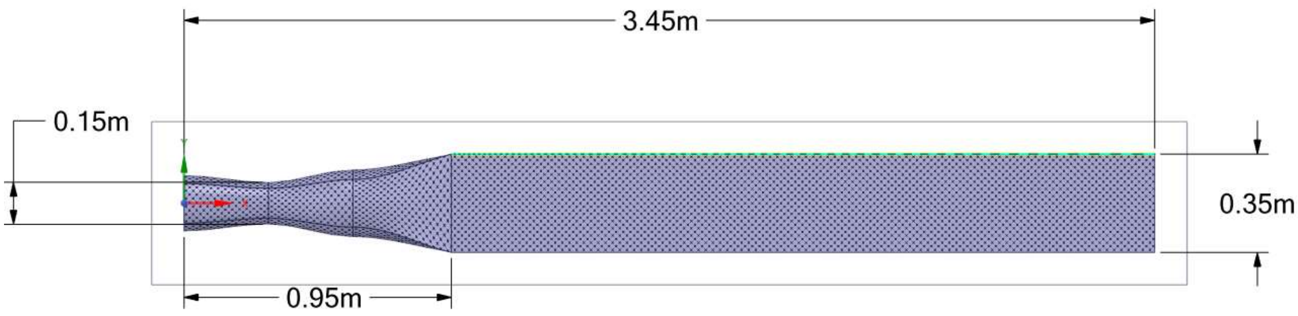
Multiple objectives optimization:

- Goal: increase **heat flux rate** and reduce **pressure drop** at the same time.
- Result: with **11** design iterations, the heat flux rate is **increased by 22%** and the pressure drop is **reduced by 9%.**

Shape Optimization Case: Wind Tunnel

Ansys

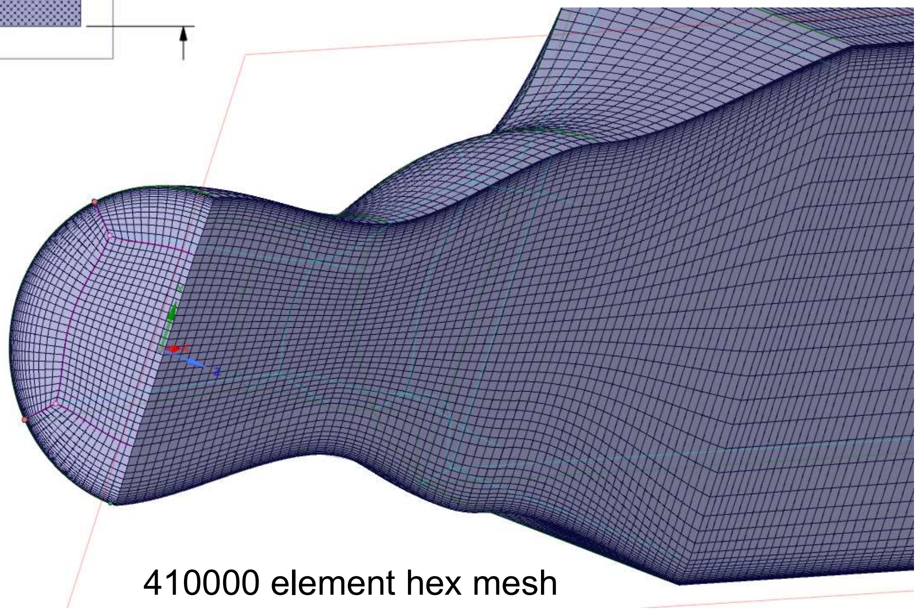
Problem Description



The wind tunnel geometry consists of a CD nozzle with 0.15 m throat diameter attached to a square duct of 0.35 m height and width.

Goals: improve the uniformity of velocity and increase the velocity magnitude in the test region by modifying the geometry of the wind tunnel walls. Use multi-objective adjoint optimization.

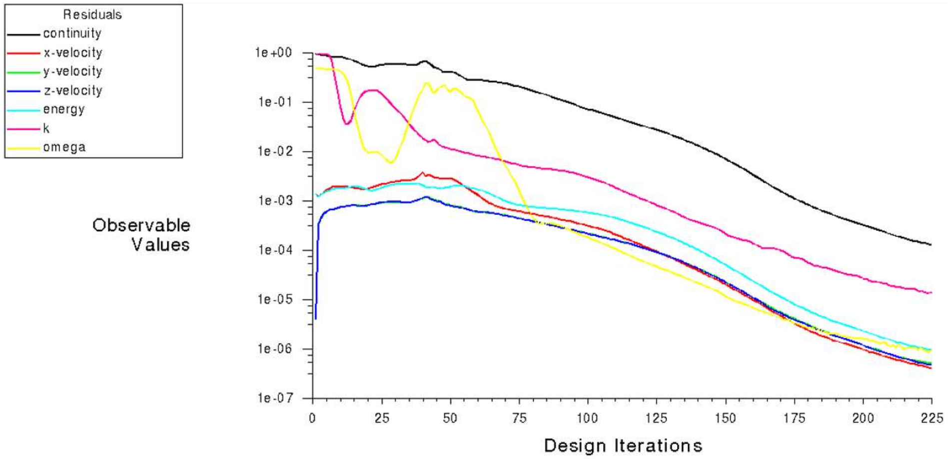
Note: the geometry is fictitious and may not be representative of a real working wind tunnel

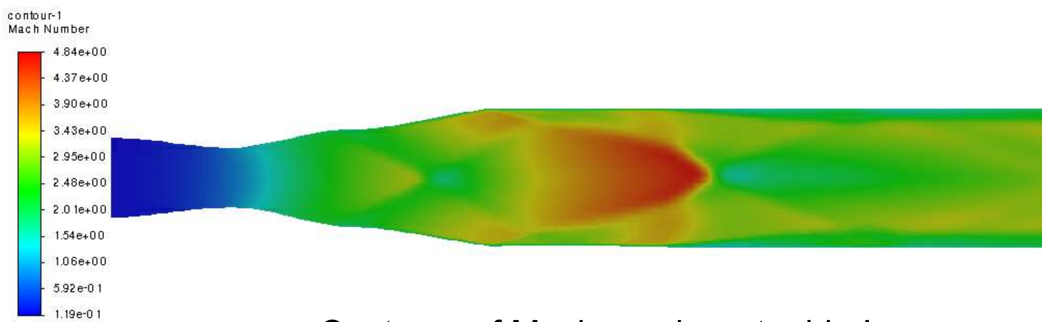


410000 element hex mesh

Baseline Fluent Solution

- General flow settings are provided below.
- Air was assumed to be an ideal gas.
- Settings
  - Pseudo-transient pressure-based coupled solver
  - K-omega SST turbulence
  - Air, ideal gas
  - Operating Pressure: 50 kPa
  - Inlet: Total Pressure 6 MPa, Supersonic/Initial Pressure 5 MPa
  - Outlet: 0 Pa

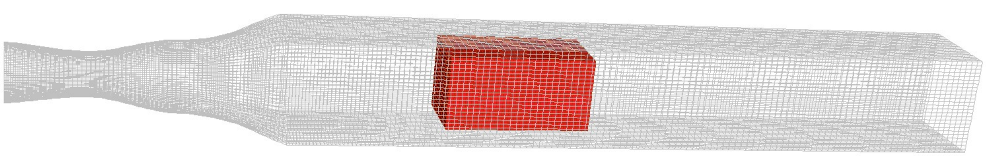




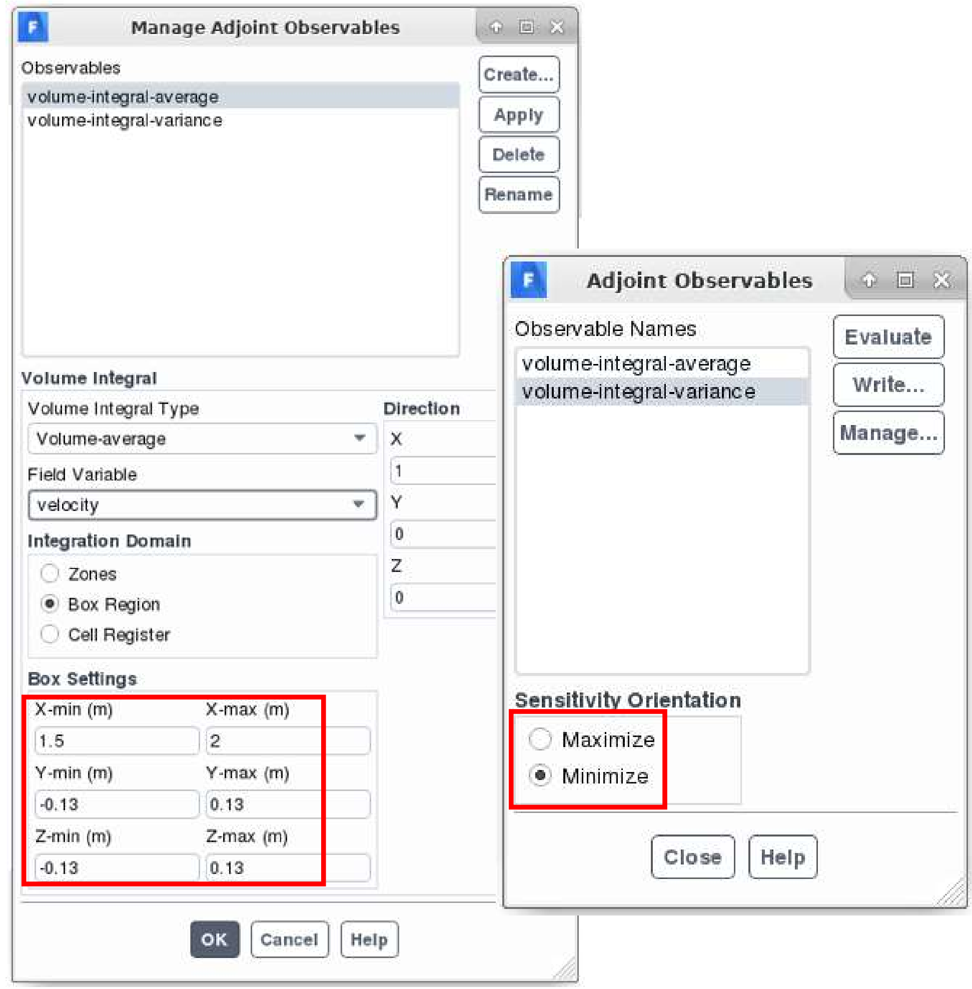
Contours of Mach number at mid-plane

Adjoint Observables

- Adjoint observable definitions
  - *Volume-average x-velocity* for the provided Box Region (below)
  - Repeat for *Volume-variance x-velocity*
  - At the Adjoint Observables window, set Sensitivity Orientation for *volume-integral-average* to *Maximize* and *volume-integral-variance* to *Minimize*



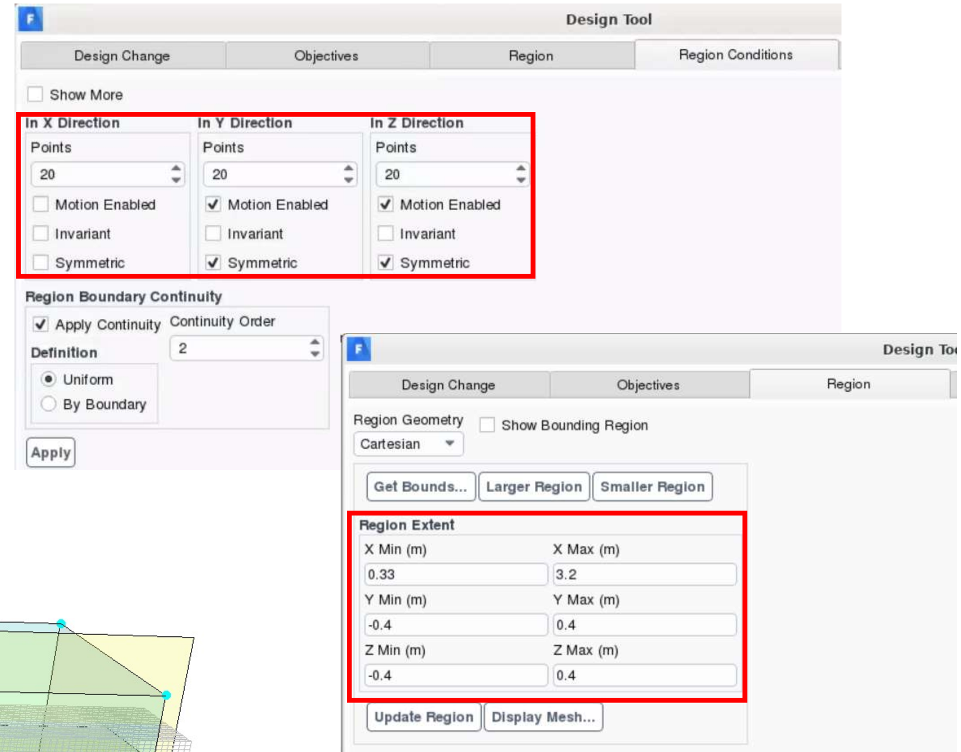
Visualization of volume integral region using cell register

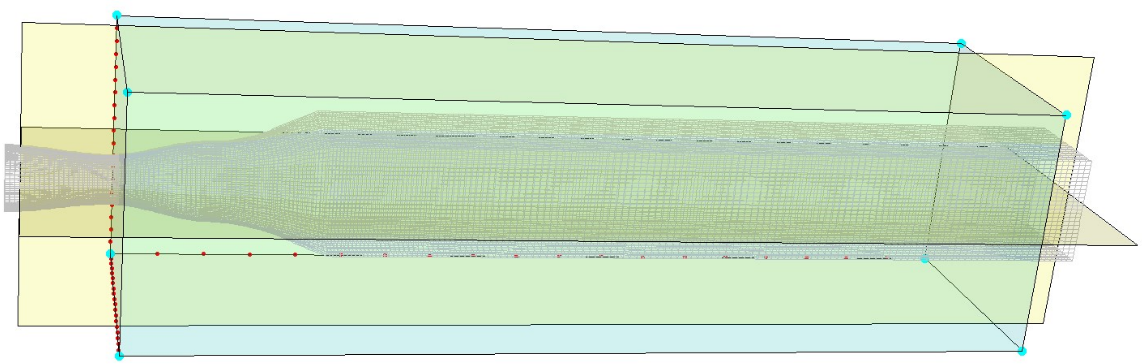


Ansys

Adjoint Design Tool

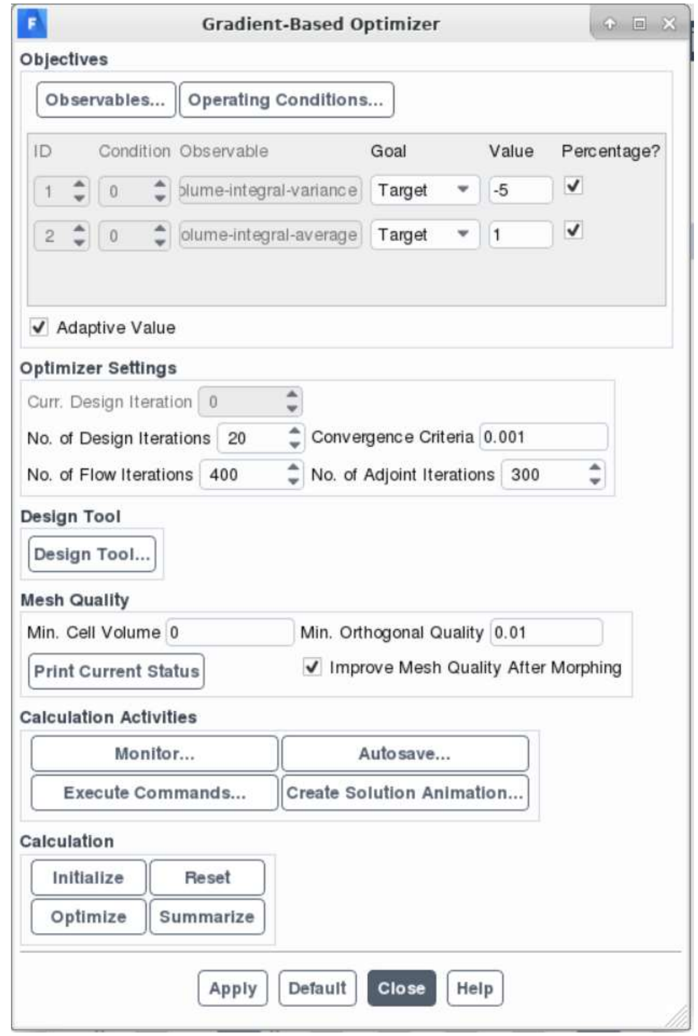
- Design Tool was used to set up basic adjoint design conditions
- Polynomial shape change method used
- Region and Region Conditions as shown
- The optimization was carried out using the Gradient-based Optimizer





Gradient-Based Optimizer

- Gradient-Based Optimizer used to perform the multi-objective optimization
- Observable target values
  - Variance decrease of velocity by 5%
  - Increase average velocity in test region by 1%
- Number of design iterations = 20
- Convergence settings as shown

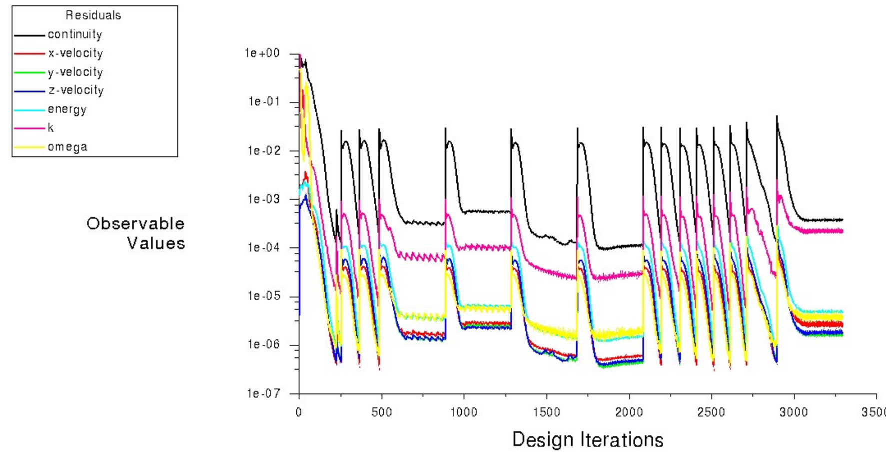


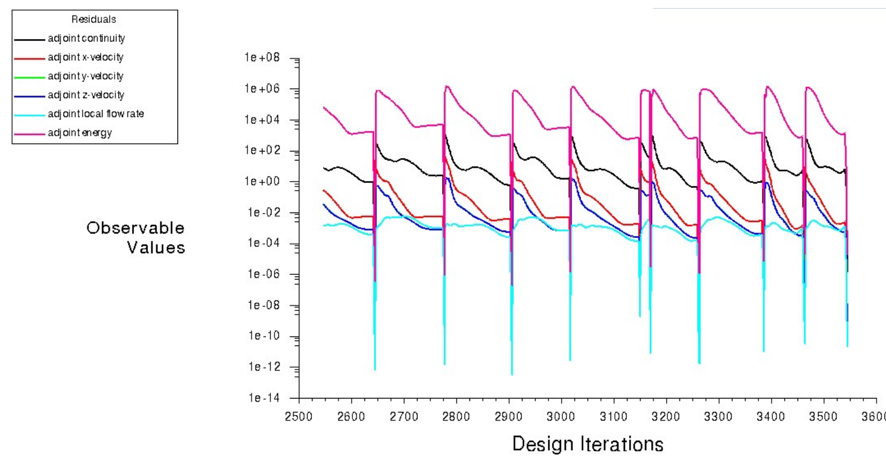
Adjoint Optimization Results

Convergence is generally good, except for a few flow calculations where the energy residuals remain above the 1e-06 threshold.

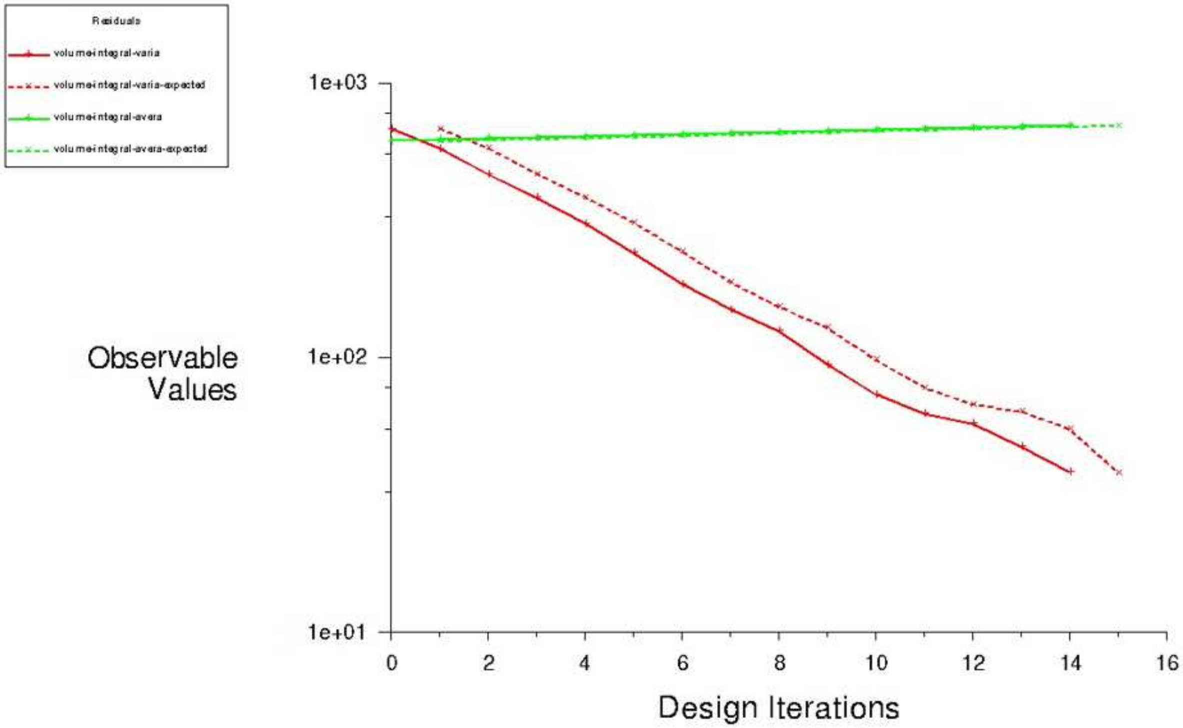
.........Summary of gradient-based optimization.....................

Design Iter.	ID	Cond.	Observable	Flow Conv.	Adj. Conv.	Observable Value	Expected Change
0	1	0	volume-integral-varia	Y	Y	6.811e+02	1.358e+00
0	2	0	volume-integral-avera	Y	Y	6.167e+02	-1.751e-01
1	1	0	volume-integral-varia	Y	Y	5.778e+02	1.537e+00
1	2	0	volume-integral-avera	Y	Y	6.226e+02	-9.542e-02
2	1	0	volume-integral-varia	Y	Y	4.652e+02	1.763e+00
2	2	0	volume-integral-avera	Y	Y	6.288e+02	4.708e-02
3	1	0	volume-integral-varia	N	Y	3.818e+02	1.979e+00
3	2	0	volume-integral-avera	N	Y	6.335e+02	1.365e-01
4	1	0	volume-integral-varia	N	Y	3.079e+02	2.187e+00
4	2	0	volume-integral-avera	N	Y	6.383e+02	2.701e-01
5	1	0	volume-integral-varia	N	Y	2.410e+02	2.510e+00
5	2	0	volume-integral-avera	N	Y	6.446e+02	5.407e-01
6	1	0	volume-integral-varia	N	Y	1.852e+02	2.870e+00
6	2	0	volume-integral-avera	N	Y	6.509e+02	9.278e-01
7	1	0	volume-integral-varia	Y	Y	1.498e+02	3.167e+00
7	2	0	volume-integral-avera	Y	Y	6.567e+02	1.224e+00
8	1	0	volume-integral-varia	Y	Y	1.249e+02	3.373e+00
8	2	0	volume-integral-avera	Y	Y	6.626e+02	1.421e+00
9	1	0	volume-integral-varia	Y	Y	9.446e+01	3.740e+00
9	2	0	volume-integral-avera	Y	Y	6.690e+02	1.884e+00
10	1	0	volume-integral-varia	Y	Y	7.339e+01	4.311e+00
10	2	0	volume-integral-avera	Y	Y	6.752e+02	2.122e+00
11	1	0	volume-integral-varia	Y	Y	6.239e+01	5.096e+00
11	2	0	volume-integral-avera	Y	Y	6.813e+02	2.191e+00
12	1	0	volume-integral-varia	Y	Y	5.746e+01	6.090e+00
12	2	0	volume-integral-avera	Y	Y	6.877e+02	2.536e+00
13	1	0	volume-integral-varia	Y	Y	4.739e+01	7.489e+00
13	2	0	volume-integral-avera	Y	Y	6.941e+02	3.419e+00
14	1	0	volume-integral-varia	N	Y	3.832e+01	-1.612e-01
14	2	0	volume-integral-avera	N	Y	6.996e+02	3.814e-01





Adjoint Results

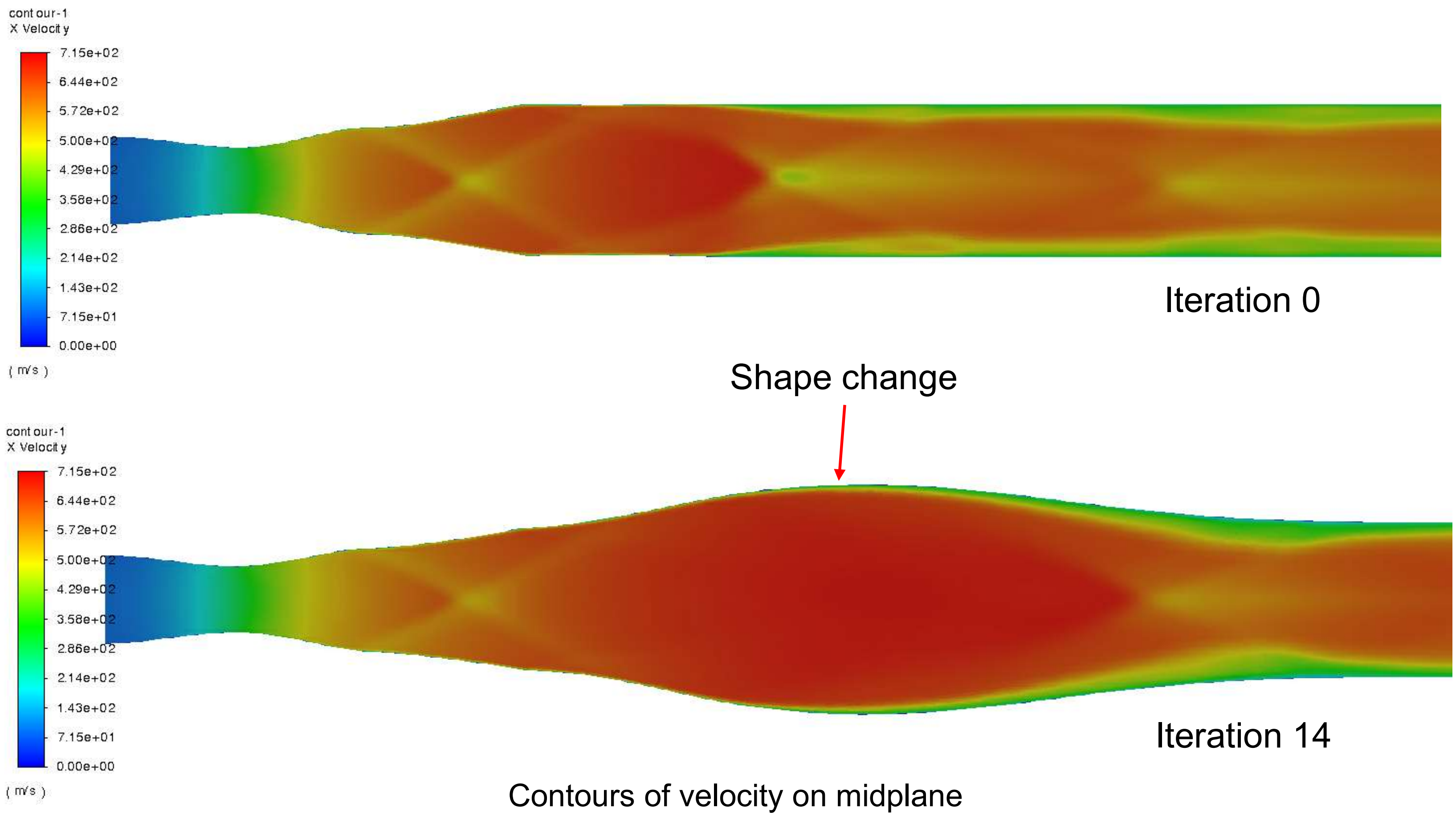


Initial
volume-average: 617 m/s
volume-variance: 681 m^2/s^2

Final
volume-average: 700 m/s
volume-variance: 38.3 m^2/s^2

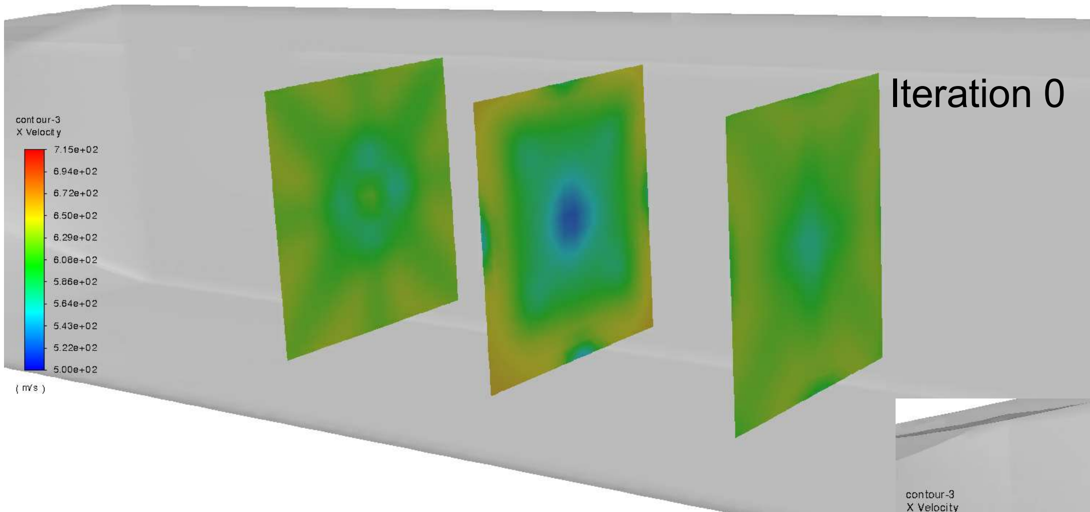
After 14 design iterations, the gradient-based optimizer was able to significantly improve the design objectives, as shown.

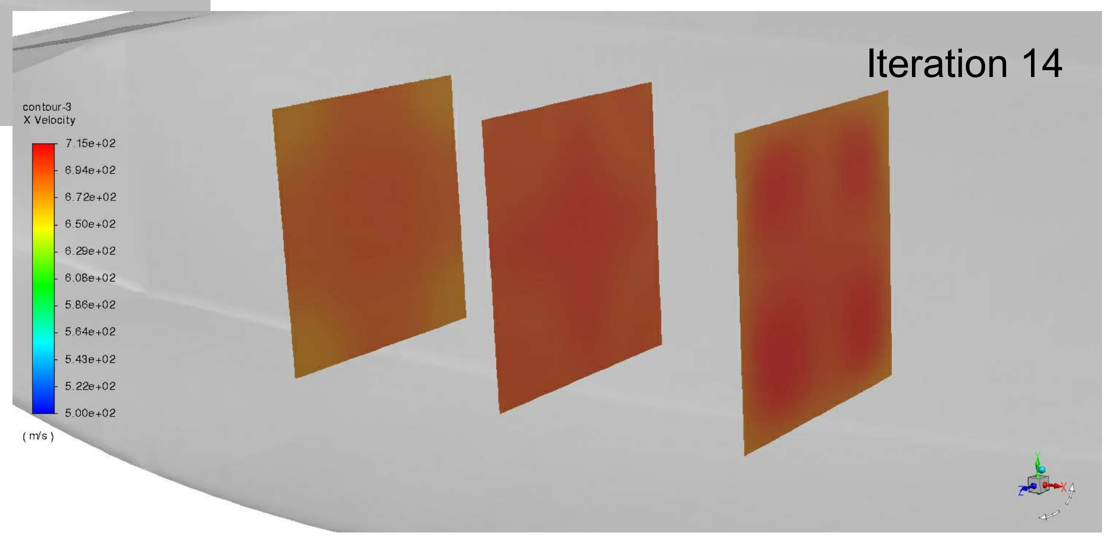
Adjoint Results – Mid-plane Axial Velocity



Contours of velocity on midplane

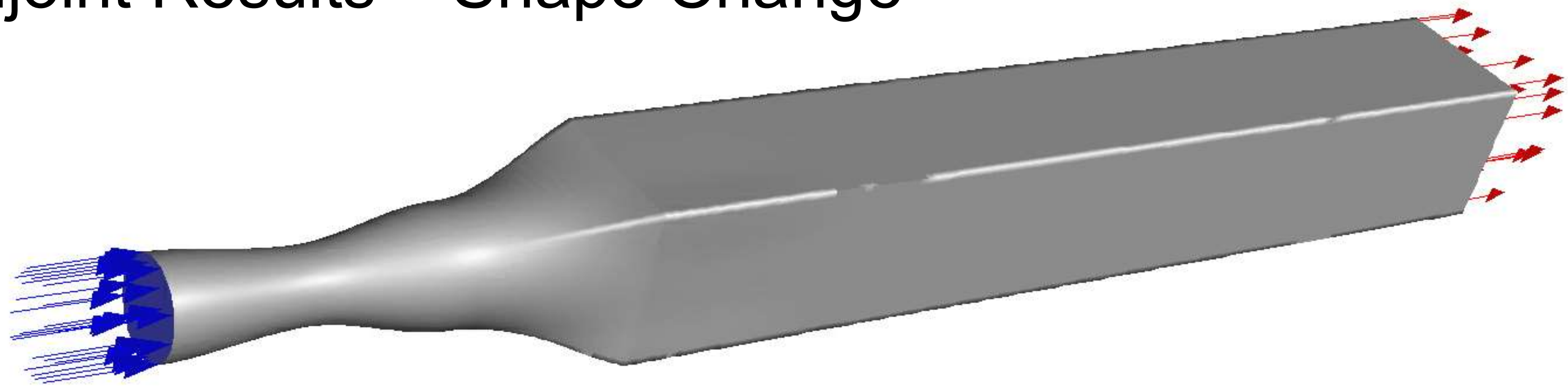
Adjoint Results – Velocity at Selected Axial Planes



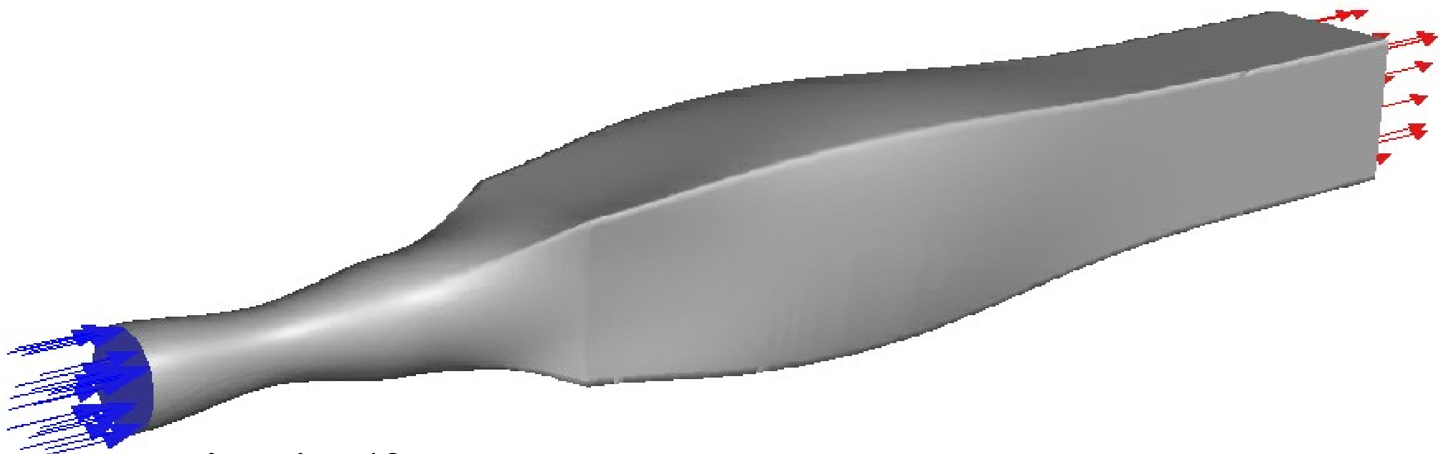


Contours of velocity on iso-clips of the volume integral region

Adjoint Results – Shape Change



Iteration 0

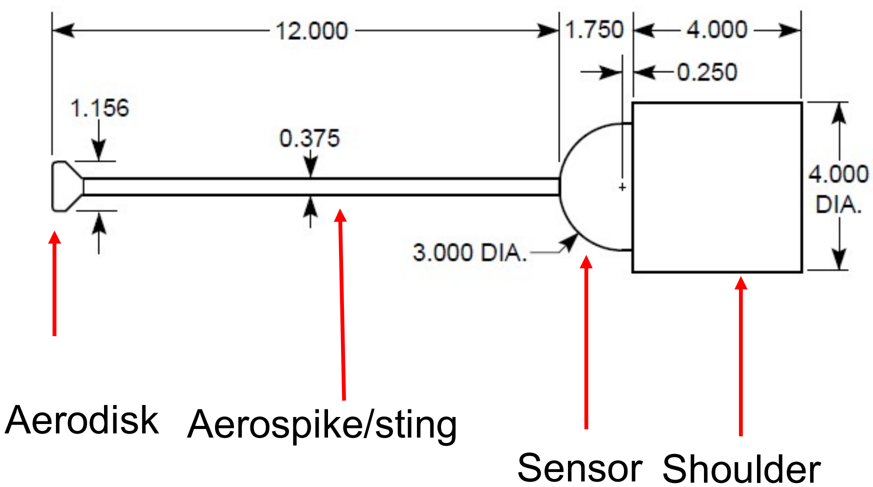


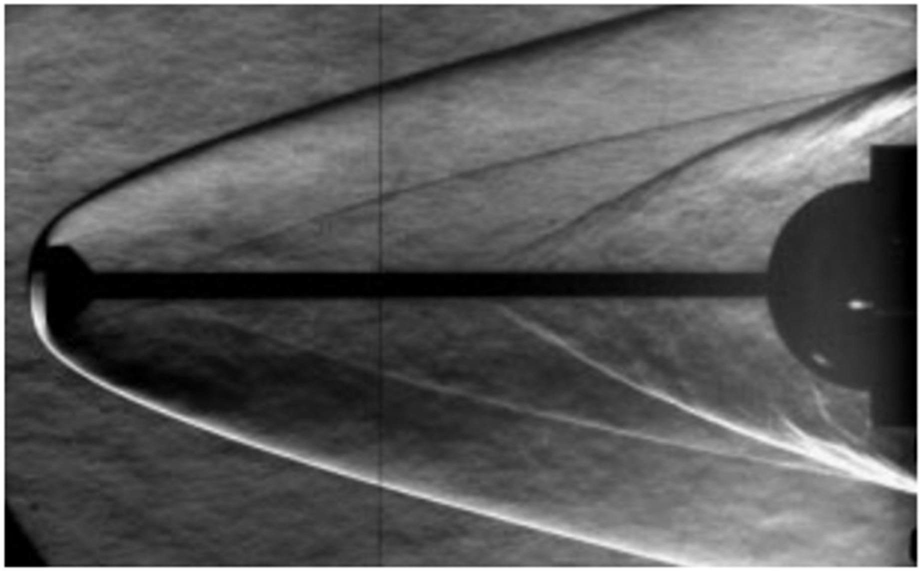
Iteration 13

Shape Optimization Case: Hypersonic Aerospike

Hypersonic Aerospike

- Present work is based on an aerospike geometry with and aerodisk proposed by Hubner et al. at NASA Langley, mid 1990s.





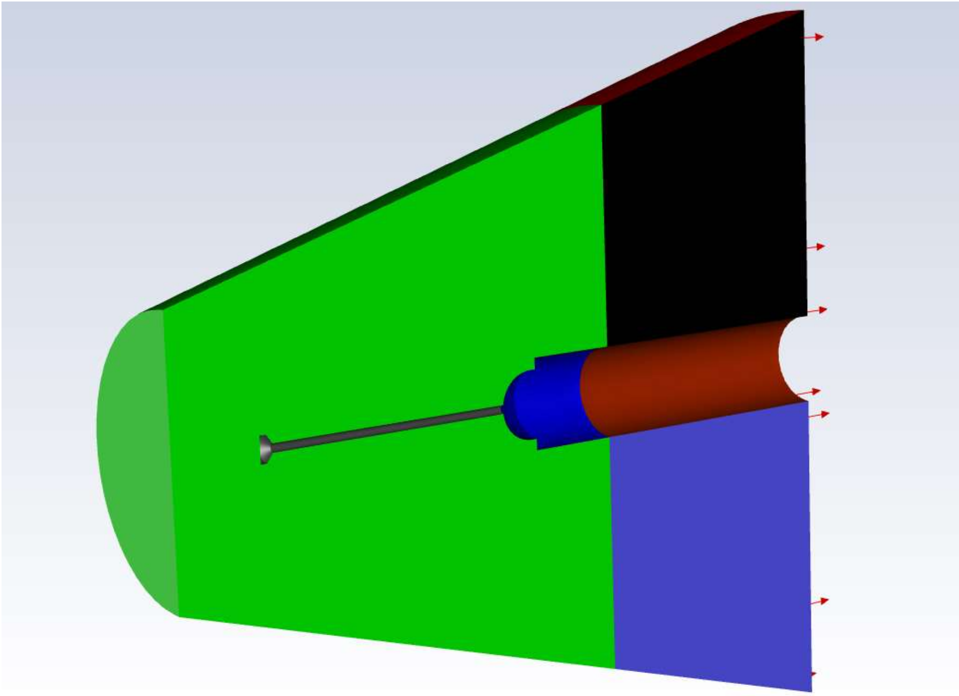
Dimensions in Inches

Reference: Huebner, L., et al., Experimental results on the feasibility of an aerospike for hypersonic missiles, 33rd Aerospace Sciences Meeting and Exhibit, Aerospace Sciences Meetings, Reno, NV, 1995.

Aerospike Optimization Case

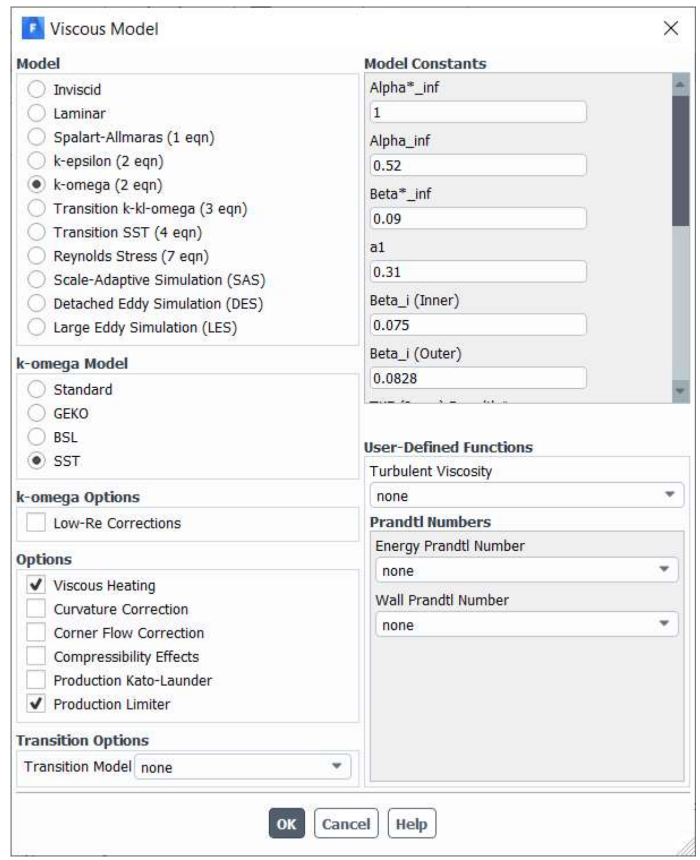
- Mach 6.06
- Upstream conditions:
  - T = 58.25 K, P = 1951 Pa
- Thermal perfect gas properties (specific heat, thermal conductivity, and viscosity are functions of temperature)
- SST k-omega model for turbulence
- Goal – Optimize aerospike to minimize drag force on the aerospike tip geometry
- Geometry – symmetric 3-D domain at zero AOA
- Mesh – 1,690,043 polyhedral cells

.



Solver Physics

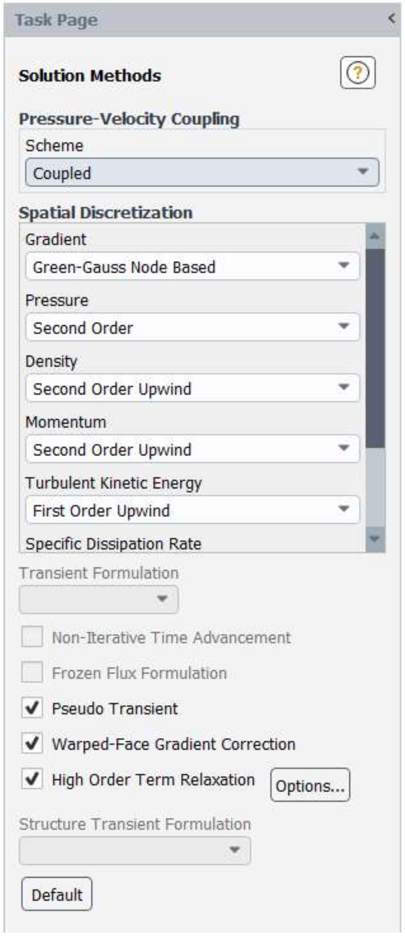
- PBNS Solver
- SST k-omega turbulence model with viscous heating enabled
- Thermally perfect air properties
- BCs
  - Mach 6 freestream conditions at inlets
  - Freestream static pressure at outlet
  - Symmetry boundary
  - No slip walls



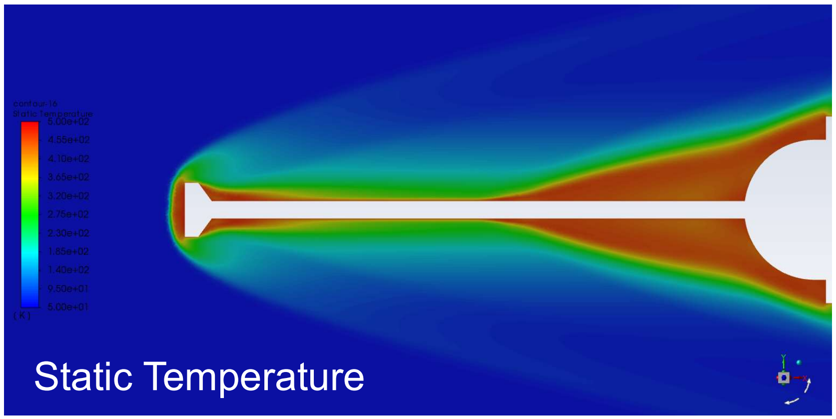
Solver Numerics

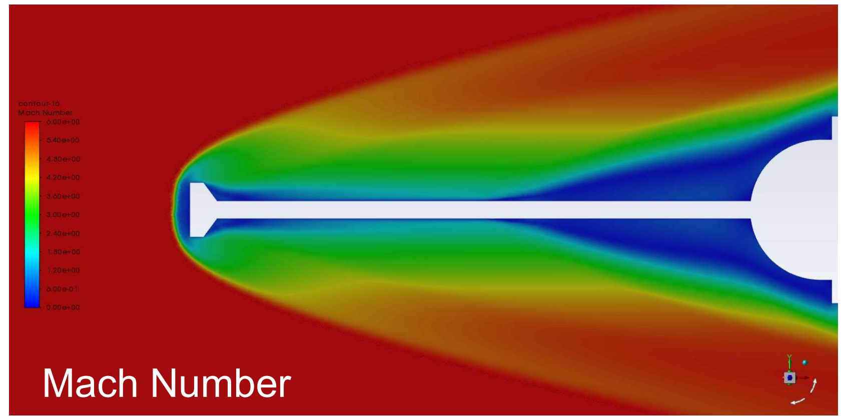
- Coupled Solver with Pseudo Transient
- Second order for all equations except turbulence
- FMG initialization
- First-to-second order blending added to provide more dissipation (TUI).
  - Improves high Mach number convergence/stability for the PB solver

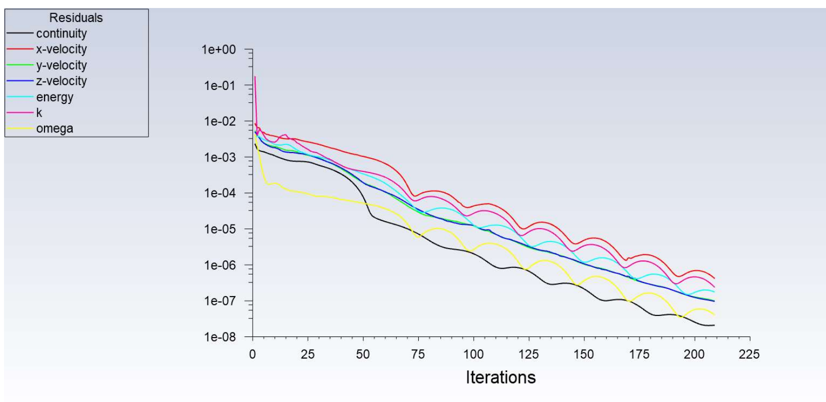
```
solve/set/numerics
.
.
1st-order to higher-order blending factor
[min=0.0 - max=1.0]: [0.7]
```

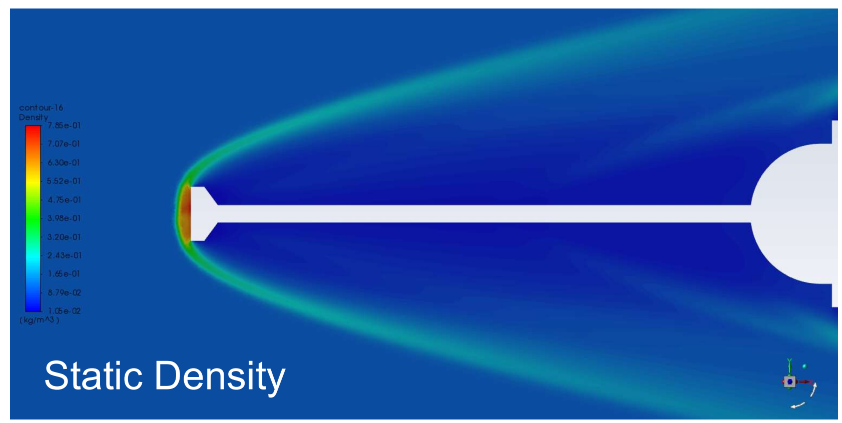


Baseline Solution

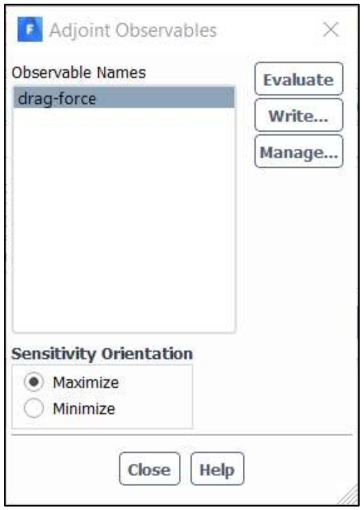


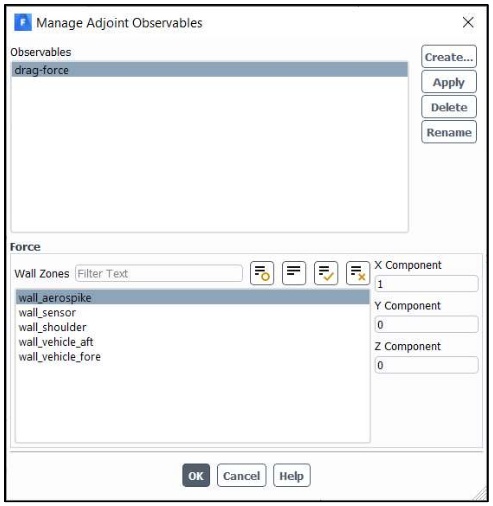




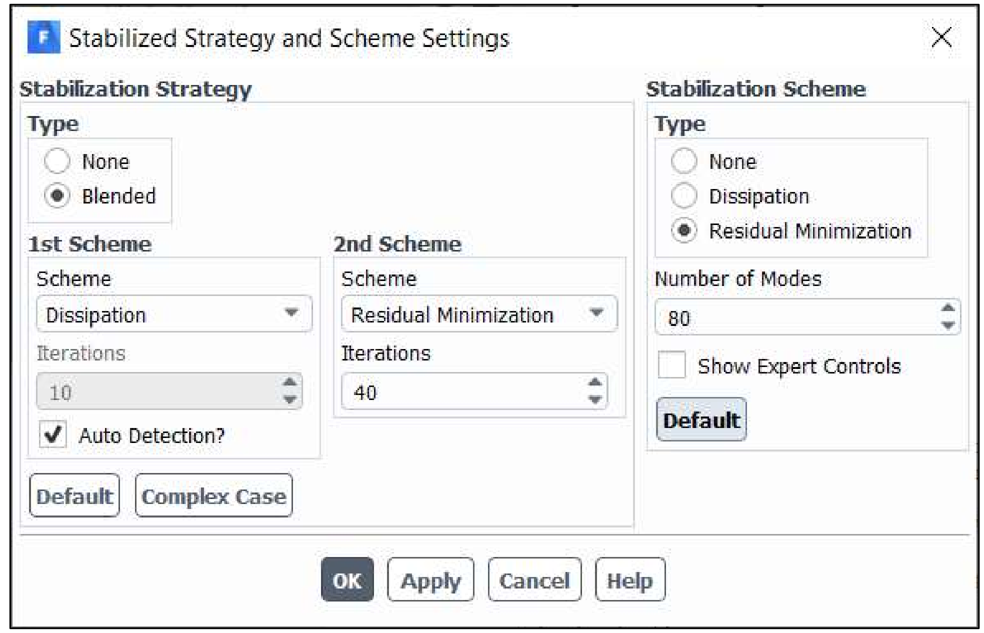


Adjoint Setup



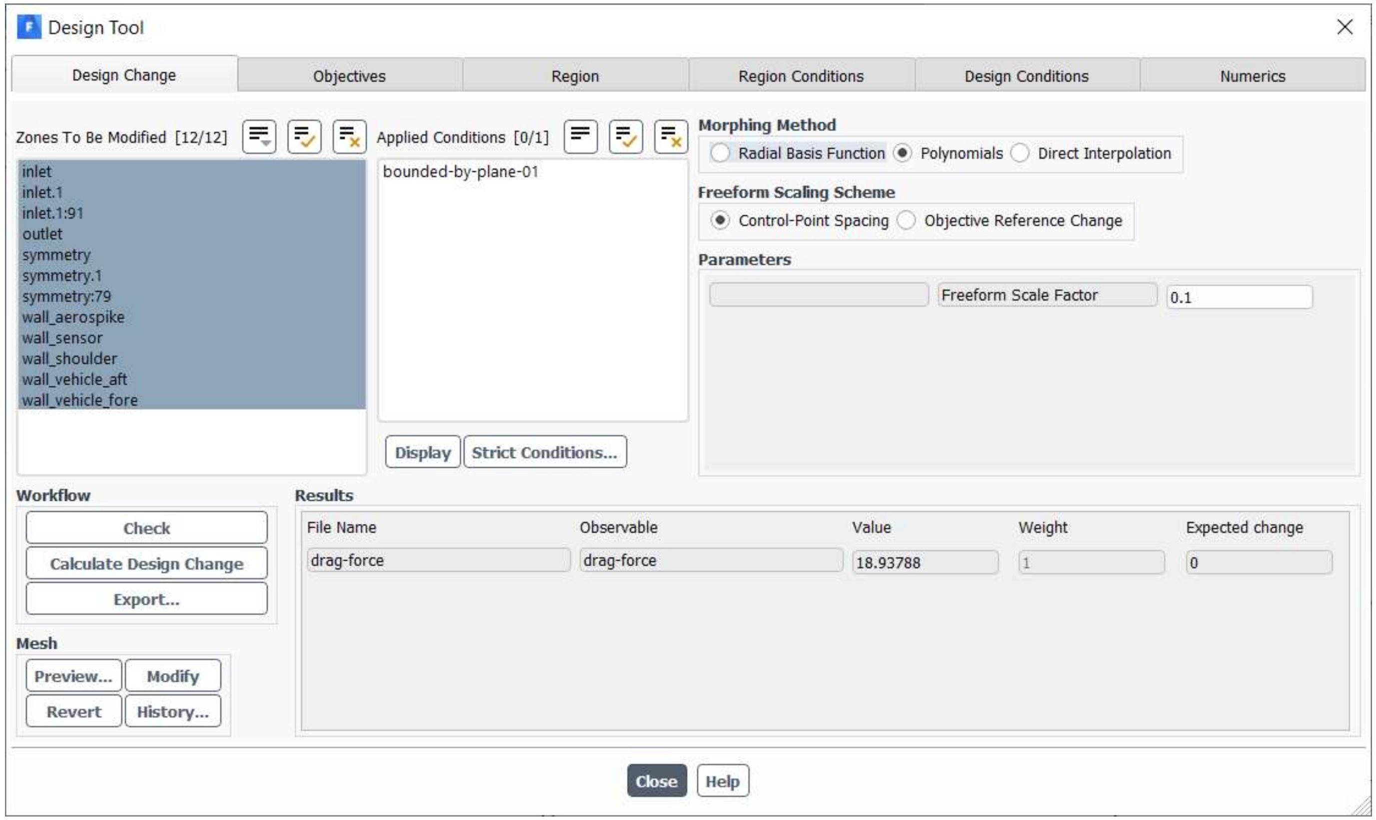


Drag force prescribed as observable

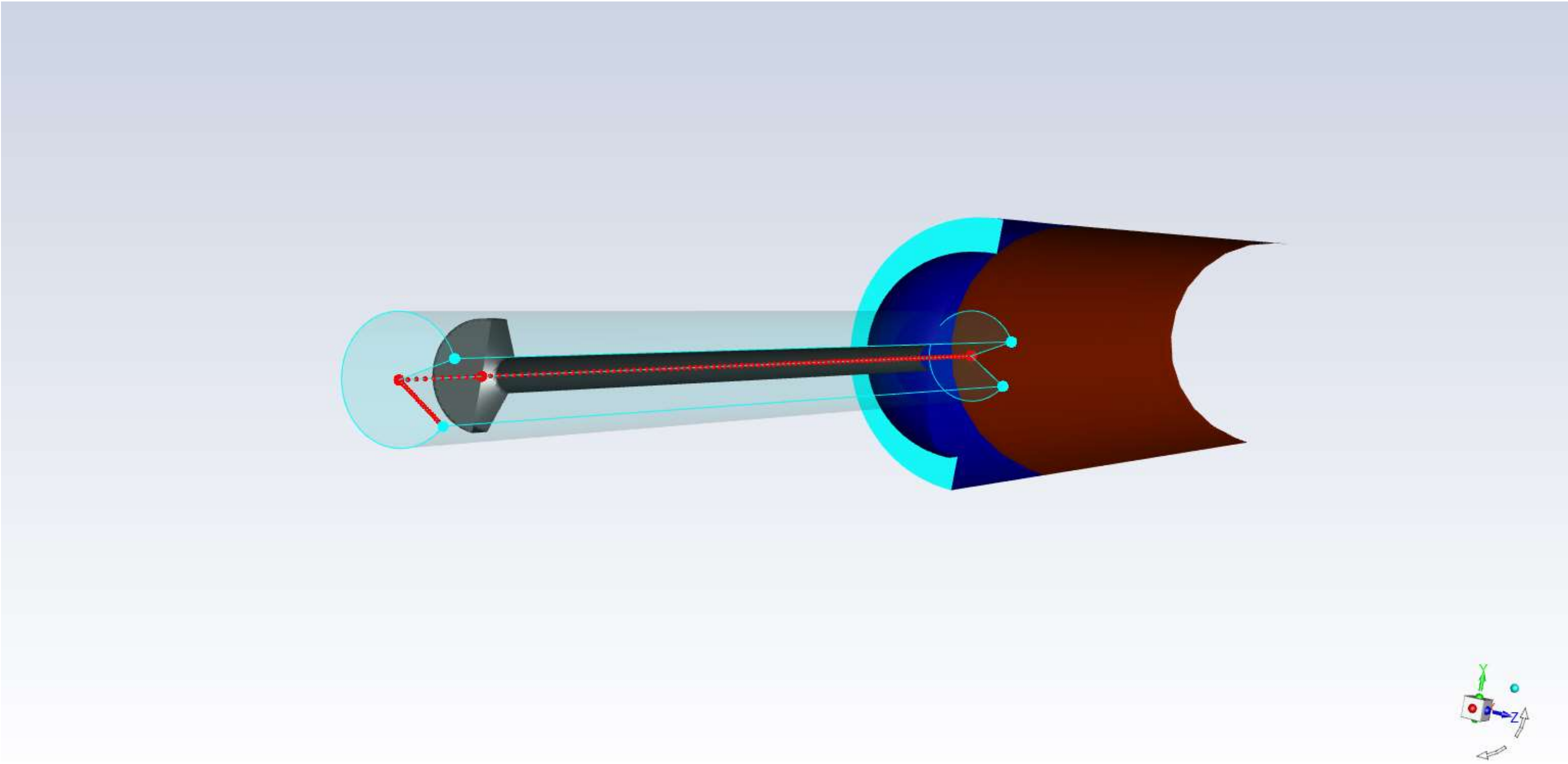


Adjoint Design Tool

- Polynomial shape change
- Shape change constrained to aerospike tip

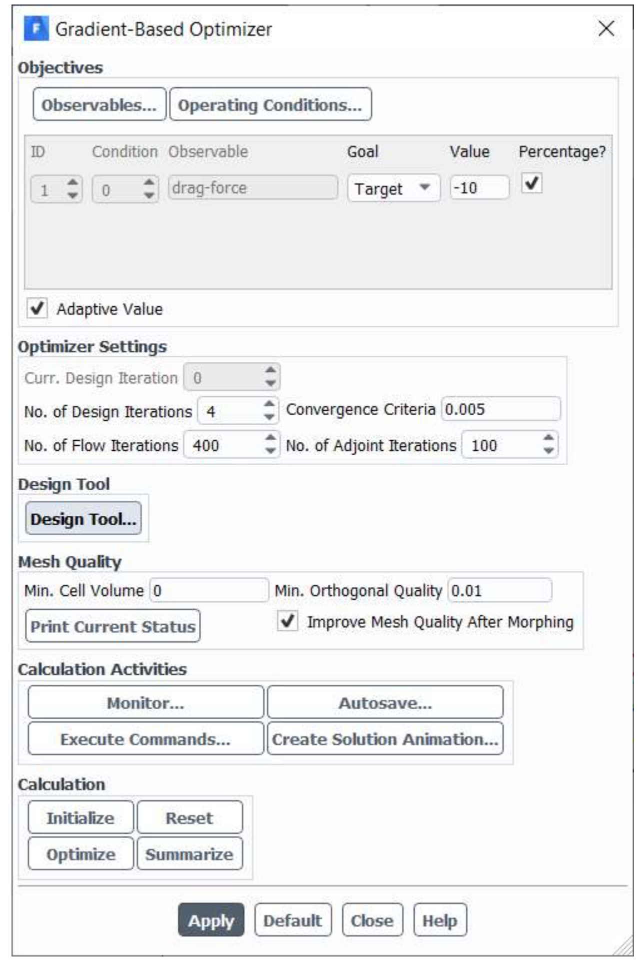


Morphing Region Definition

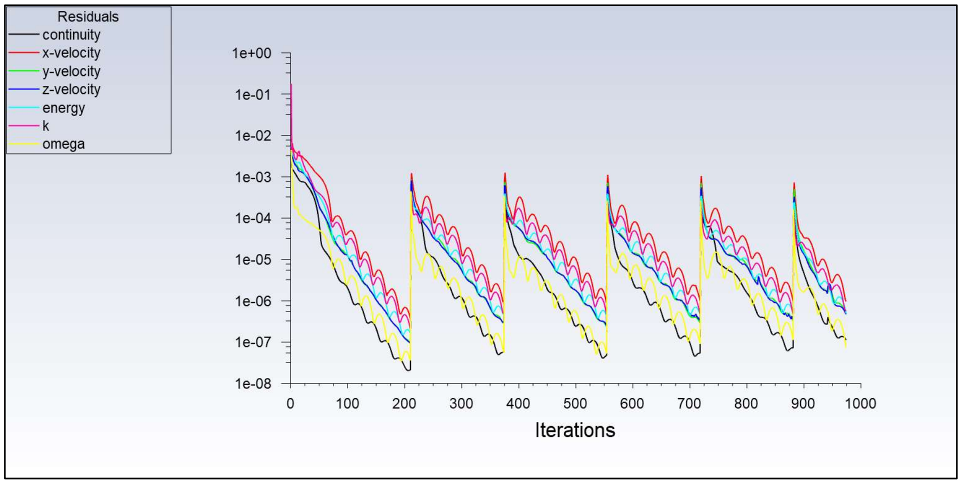


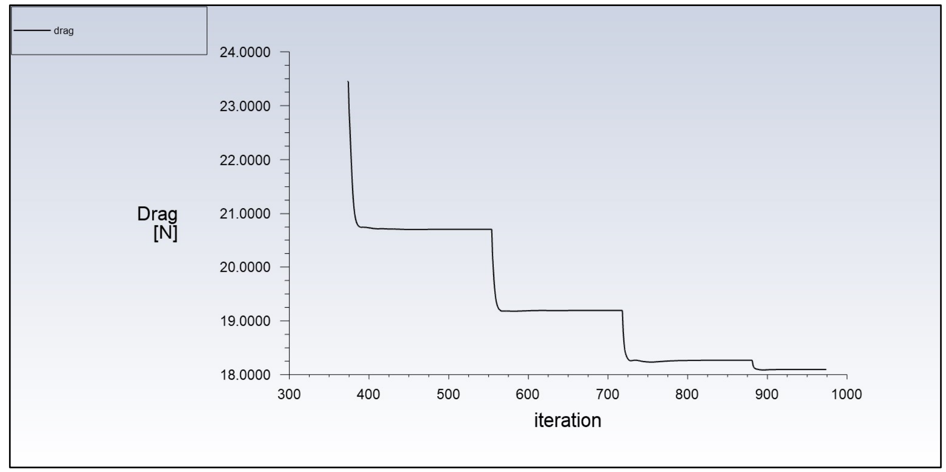
Gradient-Based Optimizer

- Single observable
- Four design iterations
- Target originally -10% reduction in drag force.
- Reduced goal at later design iteration
  - Most likely need a finer mesh at the aerospike tip to get further refinement in geometry

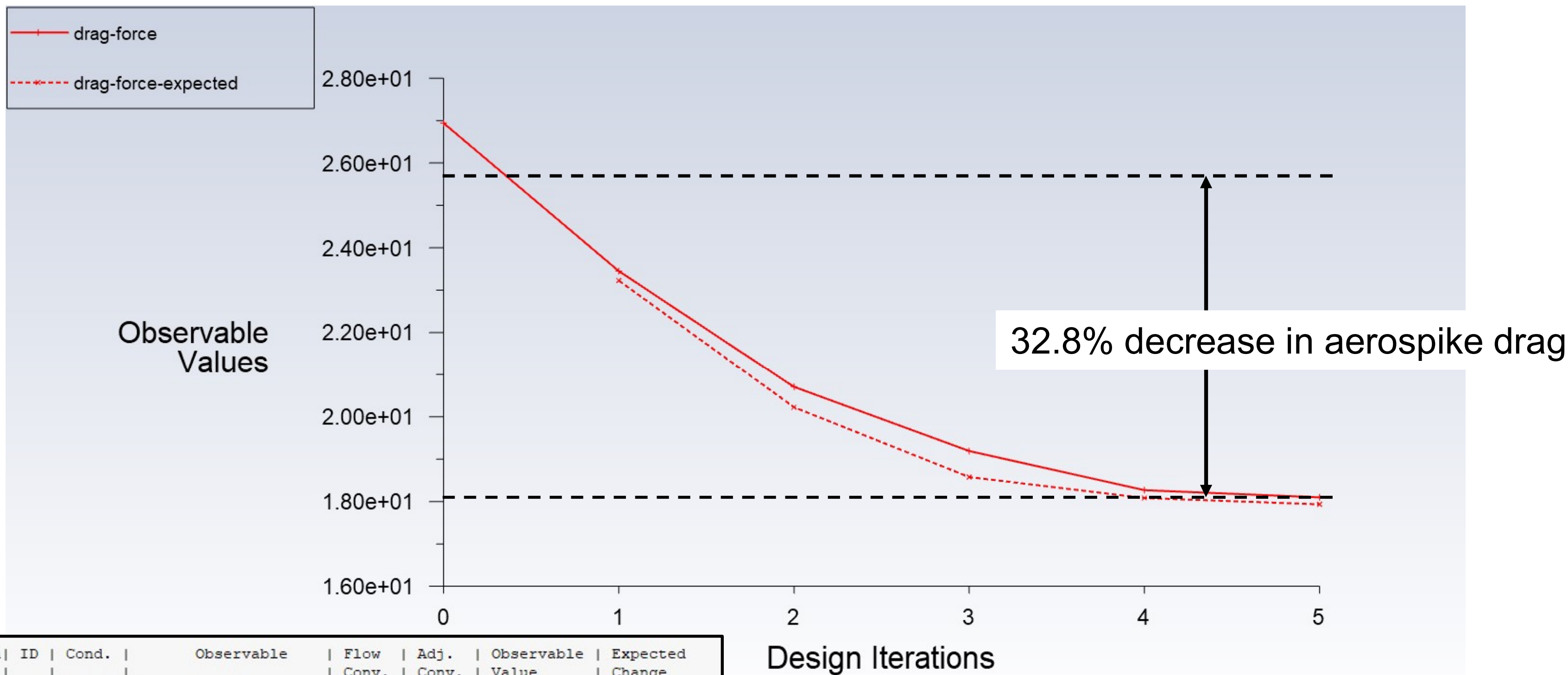


Optimization Histories



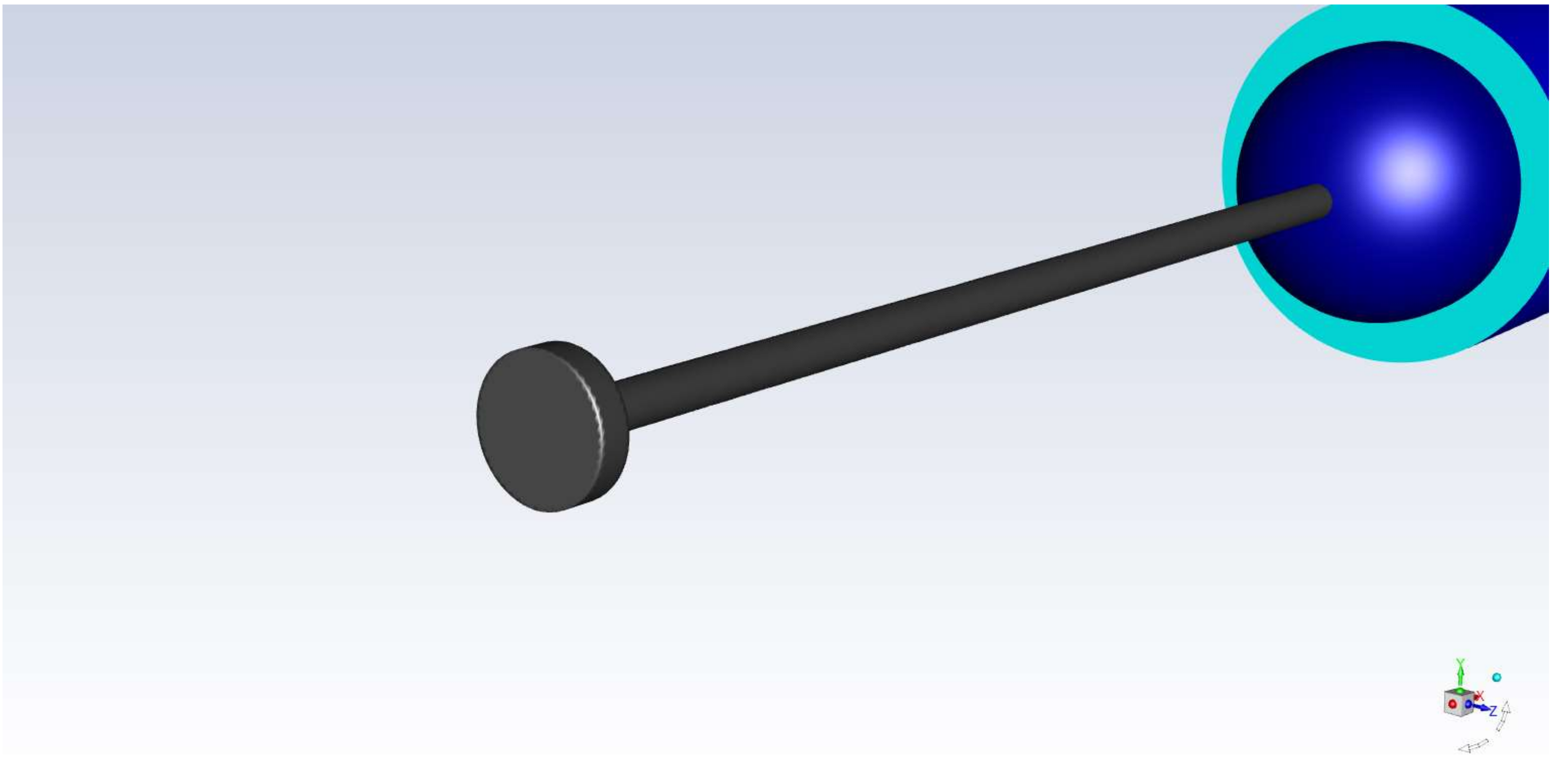


Observable History

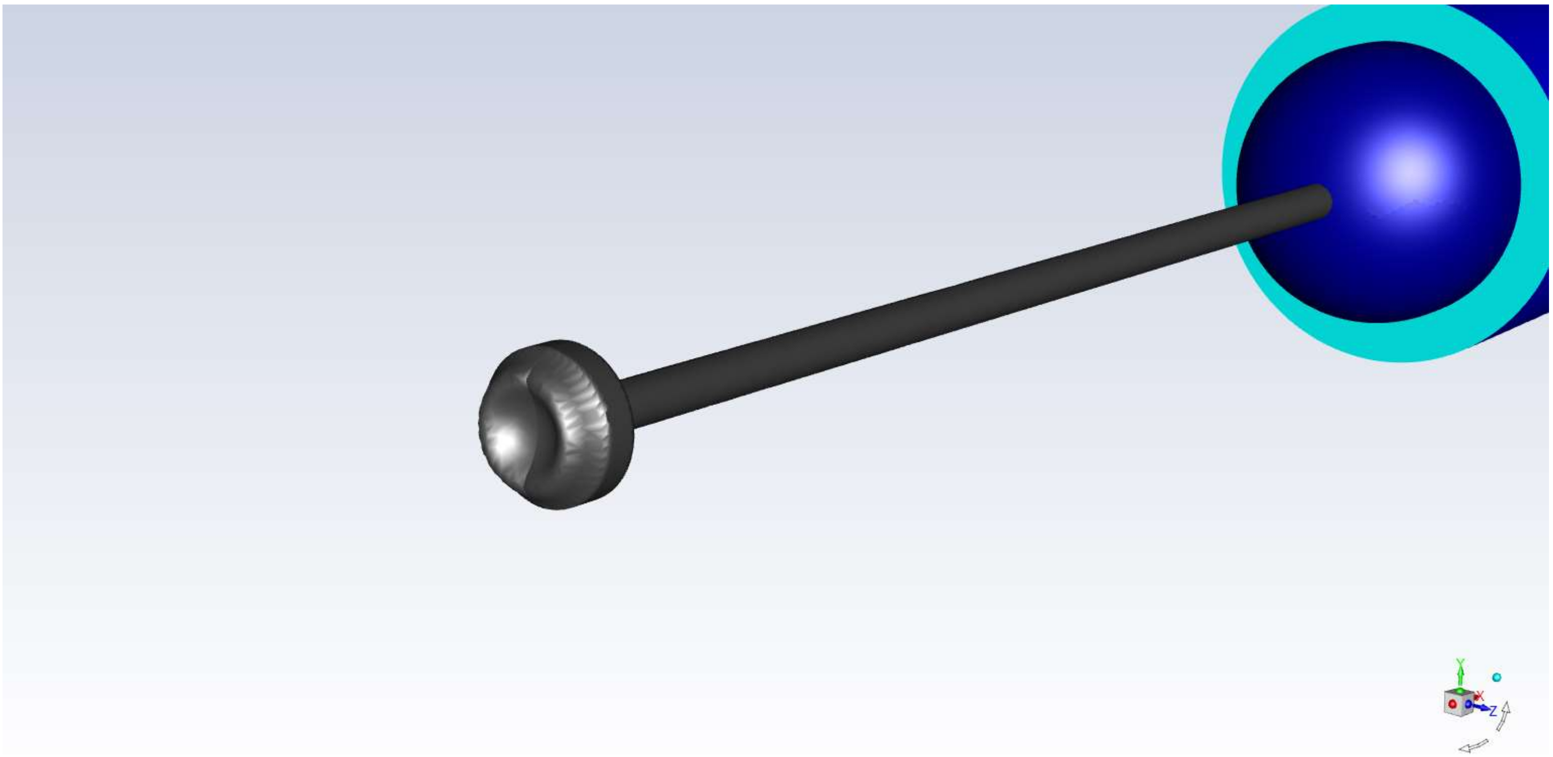


Design Iter.	ID	Cond.	Observable	Flow Conv.	Adj. Conv.	Observable Value	Expected Change
0	1	0	drag-force	Y	Y	2.694e+01	-3.715e+00
1	1	0	drag-force	Y	Y	2.345e+01	-3.222e+00
2	1	0	drag-force	Y	Y	2.071e+01	-2.131e+00
3	1	0	drag-force	Y	Y	1.919e+01	-1.109e+00
4	1	0	drag-force	Y	Y	1.827e+01	-3.348e-01
5	1	0	drag-force	Y	N	1.810e+01	undef

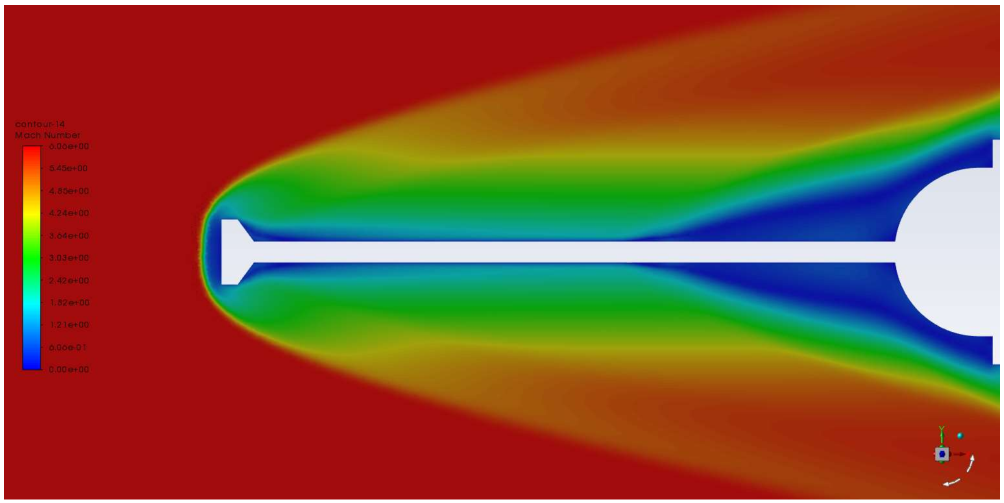
Baseline Shape



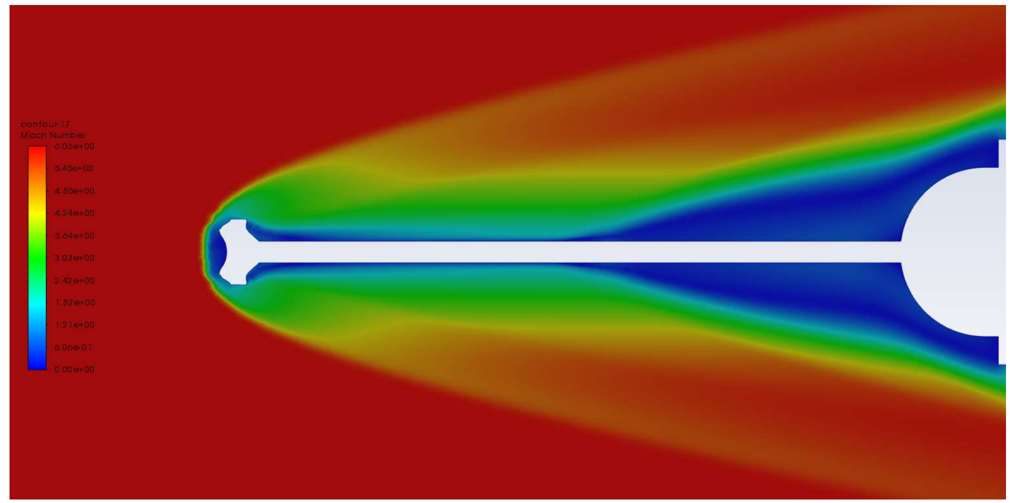
Optimized Shape



Mach Number Comparison

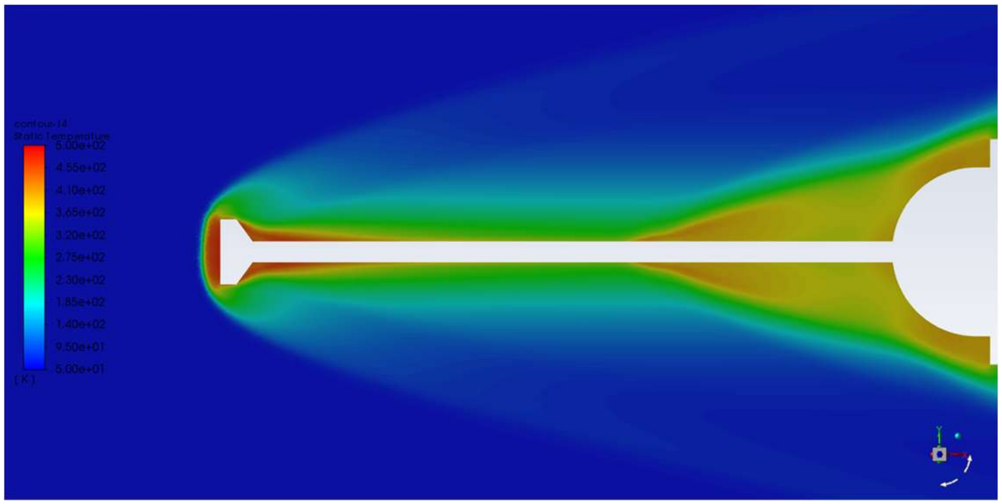


Baseline

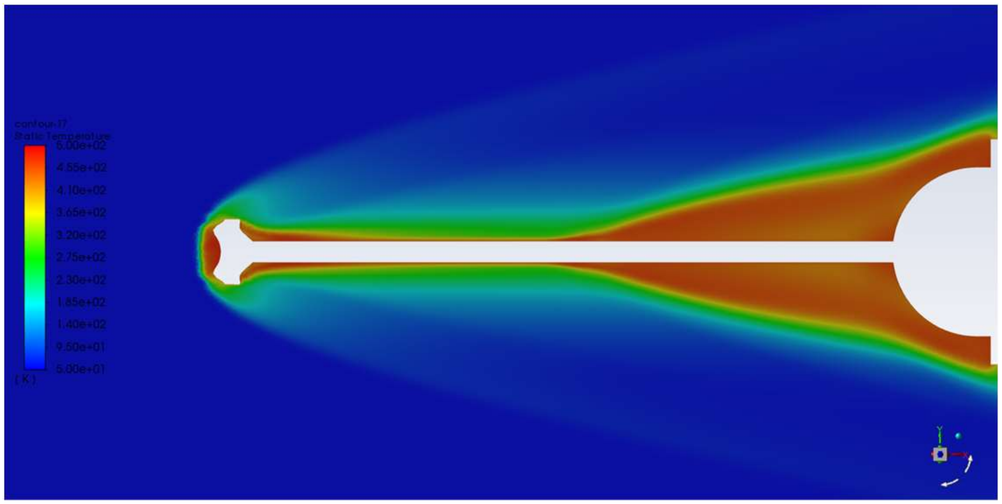


Optimized

Static Temperature Comparison



Baseline



Optimized

DBNS Force Analysis

DBNS solution for aerospike using baseline and adjoint-optimized geometry

		BASELINE					OPTIMIZED	
Zone	Pressure	Viscous	Total		Zone	Pressure	Viscous	Total
wall_aerospike	27.212	0.037	27.249		wall_aerospike	18.919	0.012	18.931
wall_sensor	7.306	-0.074	7.232		wall_sensor	8.661	-0.121	8.540
wall_shoulder	10.797	0.000	10.797		wall_shoulder	15.448	0.000	15.448
wall_vehicle_aft	0.036	1.606	1.642		wall_vehicle_aft	0.043	1.860	1.903
wall_vehicle_fore	0.003	0.644	0.647		wall_vehicle_fore	-0.002	0.858	0.856
Net	**45.355**	**2.213**	**47.568**		Net	**43.069**	**2.609**	**45.678**

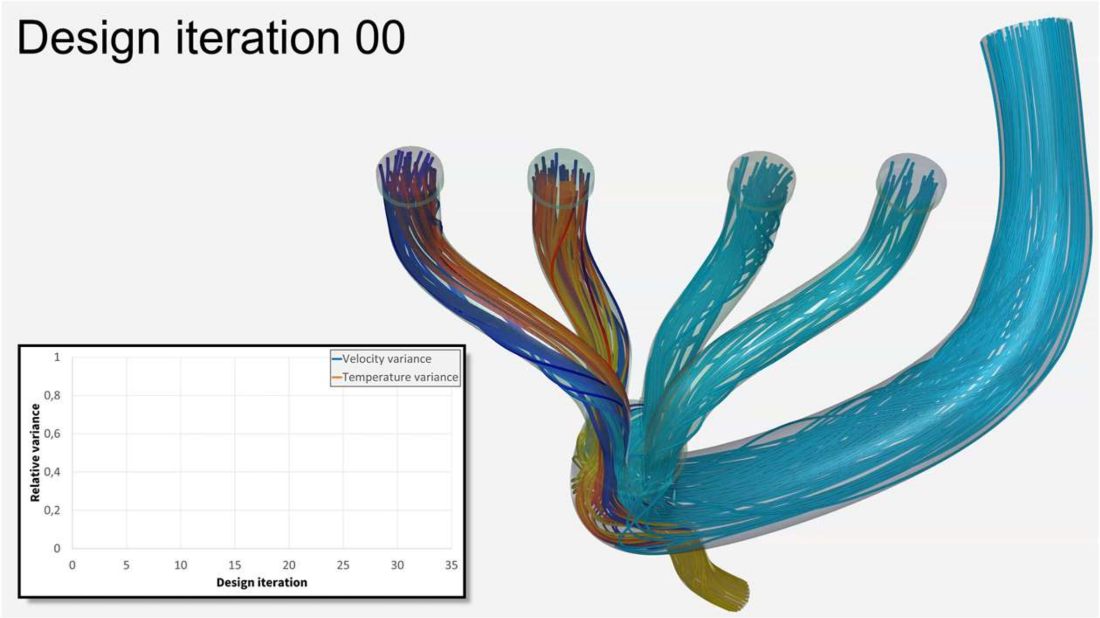
- 30.5% reduction in aerospike drag, but…
- …only 4% reduction in total drag
- Reduction in spike drag offset by increase in drag downstream…
- Nevertheless, the overall drag is reduced!

Summary

Summary: The Value of An Adjoint Approach

Understanding Cause & Effect in Complex Systems

- ✓ Insight: Identification of the most important factors affecting performance.
- ✓ Design Exploration: How a prescribed change will alter the performance. Best design change.
- ✓ Optimization: Systematic improvement of performance using gradient information.
- ✓ Robust design: Comprehensive identification of the most influential design parameters.
- ✓ Robust Simulation: Analysis of the sensitivity of the result to the mesh and numerical schemes employed.



Special thanks to Min Xu, Chris Hill, Akram Radwan, and Henry Vu for their contributions to this presentation!